BILINGUAL PUBLISHING CO.
Pioneer of Global Academics Since 1984

Journal of Mechanical Materials and Mechanics Research
https://ojs.bilpublishing.com/index.php/jmmmr

ARTICLE

Thermomechanical and Flow Analysis of a Typical Truck Radiator Using PTC-Creo

Jafar Mahmoudi[*]

Department of Sustainable Production Development, School of Industrial Engineering and Management, KTH, Sweden

ARTICLE INFO

Article history
Received: 1 May 2022
Revised: 9 October 2022
Accepted: 5 December 2022
Published Online: 30 December 2022

Keywords:
Radiator
Thermo-mechanical analysis
Flow analysis
PTC-Creo

ABSTRACT

In automobile engines, it is commonly known that the proper removal of the excess heat, resulting from internal combustion, is of high significance in the prevention of numerous negative consequences. In this regard, the radiator has a pivotal role as the main component of the engine's cooling system. Hence, its design and analysis are highly important, requiring more comprehensive failure and flow investigations. In this work, a Scania radiator is examined under the thermal and mechanical loads, followed by its analysis under the combined thermomechanical loading. Then, the flow characteristics, including the velocity, pressure, and enthalpy, are studied. In this regard, PTC-Creo software is utilized. The results demonstrate that thermal stress causes seven times more displacement than a mechanical one. When they are combined, this value reaches 1.5 mm. Also, the maximum failure index value of the Tresca theory is around 4.58, observed at the inlet side of the radiator. Besides, this paper indicates that the PTC-Creo can be considered a reliable and economical tool for the simulation of industrial applications, such as the considered radiator of a heavy-duty cooling system.

1. Introduction

The demand for more efficient and powerful engines in smaller hood spaces has caused a problem in the proper heat loss of automotive heat exchangers. As we know, in cars, the fuel and air produce power through the combustion inside the engine. Only a fraction of the total generated power is supplied to the car as power, and the rest is wasted. It is known that 33% of the energy generated by the engine is lost in heat form. If this excess heat is not removed, the engine temperature will become too high, leading to the engine overheating, which in turn causes the breakdown of the lubricating oil, weakening of the metal in engine parts, and considerable wear between the parts [1]. To minimize the pressure and stress on the engine due to heat generation, car heat exchangers must be redesigned to be more compact while maintaining a high level of heat transfer performance. A well-known type of heat

---

*Corresponding Author:
Jafar Mahmoudi,
Department of Sustainable Production Development, School of Industrial Engineering and Management, KTH, Sweden;
Email: Mahmoudi@kth.se

DOI: https://doi.org/10.30564/jmmmr.v5i2.4601

exchanger is a radiator.

A radiator is a heat exchanger used in heating and cooling systems to transfer heat energy from one medium to another. Most radiators are designed and built to be used in cars and buildings and usually function as a water temperature carriers. Figure 1 shows a typical truck radiator.

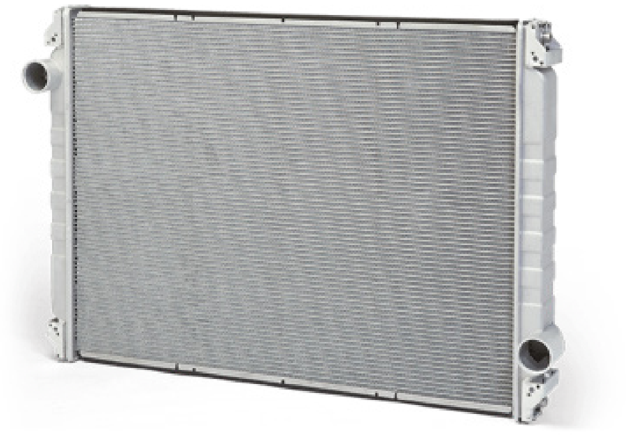


Figure 1. Typical Truck Radiator

Their structure is such that the liquid circulates in their pipes and causes the environment to heat or cool [2,3]. They are made of different materials, including aluminum, steel, copper, and cast iron. In terms of heat transfer, copper can be said to have the best transfer, but due to its very high cost, it is not economical to use. After copper, aluminum has the best heat transfer that does not rust. Most modern cars use aluminum radiators. These radiators are made by soldering thin aluminum fins to the flattened aluminum tubes [4]. The coolant passes through many tubes mounted parallel from the inlet to the outlet. The fins conduct the heat of the tubes and transfer it to the air passing through the radiator. Therefore, proper analysis of the radiator is vital in order to accomplish high-performance designs.

Numerous research works have been done on the radiator of automobile engines in various aspects. Aravindkumar et al. [5] examined a radiator tube with varying geometries using different nanofluids. The geometries were the straight and spiral tubes, designed with Catia, and the nanofluids included, for example, beryllium, oxide, copper oxide, silicon carbide, and tin oxide. They depicted the contours of static pressure and velocity vectors and then concluded that the spiral flow and the copper oxide are the best choice for the radiator's geometry and nanofluid, respectively. Sheikhzadeh et al. [6] investigated the thermal performance of a car radiator using Ethylene Glycol/copper nanofluid in various environmental conditions. They showed that increasing the volume fraction values of nanoparticles and Reynolds number of the inlet air leads to a increase of the heat transfer rate. Besides, adding nano-sized particles to the radiator coolant fluid can significantly reduce its output temperature. Moreover, a review work is presented on the application of nonfluids in the automotive radiator by Maysam Molana [7]. Another similar work has been recently done [8], addressing the hybrid nanofluids.

Rinu Sathyan [9] compared the ordinary straight tube of the radiator with the helical type. The author used SolidWorks for modeling and ANSYS for fluid flow analysis. He provided the temperature distribution for different mass flow rates and concluded that the proposed design is preferable to the straight type due to the better performance and size. Chidley et al. [10] carried out the thermomechanical analysis of automotive heat exchangers using Comsol. They used InfraRed thermography to record the thermal history of the radiator, along with the adoption of an energetic fatigue criterion. Similar work is done by Roger and Chidley [11] as well. Gu et al. [12] addressed the tank leakage of aluminum alloy radiator due to corrosion. Baou et al. [13] tried to enhance the radiator performance using different porous fin configurations and materials. They resulted that the corrugated pattern provides the best thermal performance among the considered geometries, and the horizontal configuration leads to the lowest pressure loss. Also, it was demonstrated that using porous media in the radiator channels improves its overall thermal performance factor by up to 237%. The resulting technical drawing is shown in Figure 2.

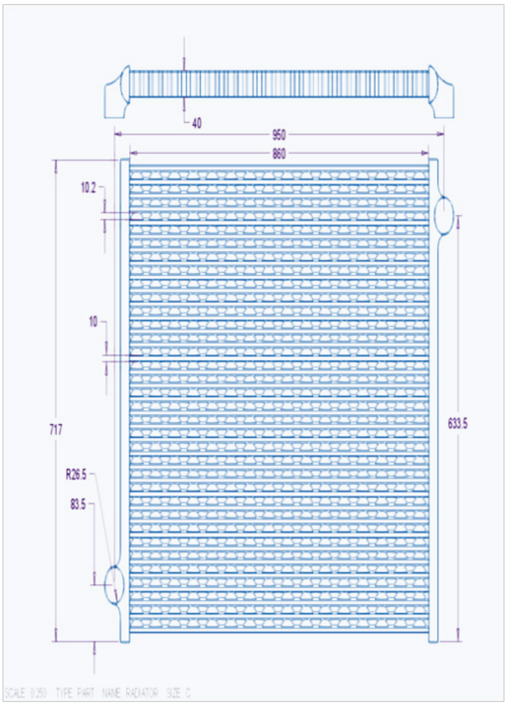


Figure 2. Technical drawing geometry [22]

Kim et al. [14] proposed a numerical method for the determination of temperature distribution in automobile radiators without requiring a prototype model. Amrutkar et al. [15] performed the theoretical, numerical, and experimental heat transfer analyses of a radiator. They used ε-NTU method to describe the heat transfer calculations and to determine the radiator size. Sahriff et al. [16] studied the fins' effects on the car radiator performance under the atmospheric temperature of Kano State of Nigeria. Honda Civic 2000 car radiator was modeled in SolidWorks and simulated in ANSYS with water as the cooling fluid. Their results showed a 25% decrease in the outlet temperature in the case of attaching fins to the radiator. Regarding energy dissipation, the finned radiator dissipates 74% of the cooling energy, while the radiator without a fin dissipates only 40.8%. Mia and Hossen [17] provided the CFD analysis of a radiator tube with fin using ANSYS FLUENT. The authors represented the variations of pressure drop, temperature difference, and velocity contours. They observed the great effect of the volume fraction of nano particles and Reynold number on the preferred temperature and pressure.

Patel et al. [18] carried out the heat transfer analysis of a car radiator with different mass flow rates of the coolant using an ANSYS workbench. Kasu et al. [19] designed a radiator using SolidWorks, performed CFD analysis and compared it with experimental results. In addition to the above research works, there are two studies examining the radiator in a different way. Marc Bonato [20] studied a brazed aluminum alloy car radiator undergoing thermomechanical loadings. He intended to determine the effect of the manufacturing process on the durability of the component in the field. The author concluded that the brazing process is well-controlled in all production units and for different radiator technologies. Goyal et al. [21] researched the reliability measures such as the availability, reliability, and MTTF of the automotive water cooling system considering the time and its failure rate parameters using Markov process and supplementary variable technique. Also, they investigated the sensitivity of these reliability characteristics to enhance the functioning of the water cooling system.

While the above-mentioned references have made valuable contributions, the present work aims to provide three main contributions: (1) simultaneous analyses of thermal, mechanical, and thermomechanical with the determination of the failure index based on the Tresca theory, (2) presentation of the flow characteristics including the velocity, temperature, and enthalpy, (3) using PTC-Creo as both CAD and CAE software. To the knowledge of the author, the utilized software has not been used for the analysis of car radiators, and there are limited papers examining the capability of this software in industrial applications. For this purpose, initially, the modeling procedure, including the thermal, mechanical, and thermomechanical analyses, is explained in Section 2. Then, the related results and discussions are provided in Section 3.

An internal combustion engine (piston engine) is a type of heat engine whose task is to convert supplied heat into mechanical energy [22]. When the fuel is burned, heat is developed which is absorbed by gas enclosed in a cylinder. When the gas is heated, the pressure rises, which in turn pushes the piston outwards and thus mechanical work is performed.

Very high temperatures occur in the combustion chamber, between 1800 °C to 2200 °C [22]. Therefore, cooling of the cylinder walls, cylinder head, piston top, etc. is required. The gas temperature is higher in otto engines (gasoline engines) than in diesel engines. About 20%~30% of the amount of heat supplied by the fuel is removed by cooling.

The cooling can be done by either water or air cooling. As air has a significantly lower cooling capacity than water, air cooling only occurs in small engines and aircraft and motorcycle engines. With water cooling, water circulates in a closed system through the engine's cooling channels. Cooling water absorbs heat that is emitted to the surroundings via the radiator itself.

2. Modeling Procedure

In this section, the procedure to analyze the chosen radiator, as shown in Figure 3, is explained. The First step is to make an accurate CAD file for the FEA simulation, which is carried out in the PTC-Creo Part environment.

The second step is the material assignment. Aluminum alloys are utilized widely for radiators [24]. While Aluminum 1050 has a higher thermal conductivity value (229 W/m*K), it has weak mechanical properties [25]. To get better mechanical feedback, the 6xxx series of aluminum alloys are commonly used, and their thermal conductivity values depend on the temperature of the alloy. Also, in general, accurate material behavior is dependent on the strain rate, temperature, material texture, and so on. In this work, some assumptions are applied to simplify this behavior. The material is supposed to be isotropic with a linear stress-strain response to the forces. Two different failure formulations are selected to discuss as well. Table 1 shows the specifications of the considered material.

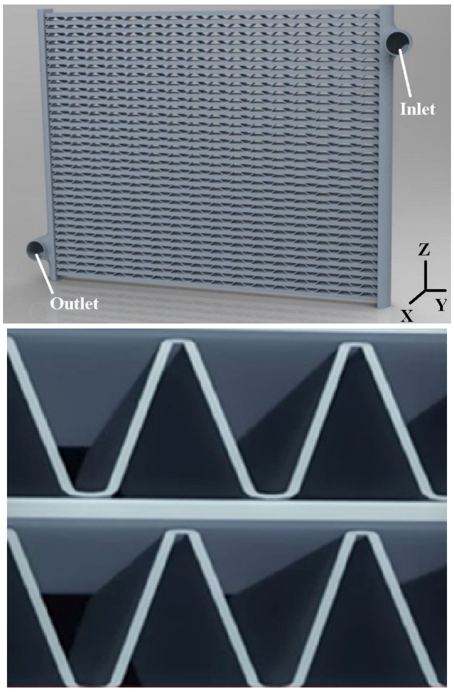


Figure 3. Simulation Domain Radiator [23]

Table 1. Material properties of the wrought aluminum [25]

Specification	Value	Unit
Poisson's ratio	0.33	-
Young's Modulus	7.05e+07	KPa
Thermal expansion coefficient	2.35e-05	1/°C
Specific heat capacity	8.98e+08	mm^2/(s^2 °C)
Thermal conductivity	229000	mm kg/(s^2 °C)
Density	2.71e-06	Kg/mm^3
Tensile ultimate strength	290	MPa
Tensile yield strength	185	MPa

The third step is to define the type of analysis. In the current study, three simulations, including thermal, mechanical, and thermomechanical analysis of the radiator, are discussed.

General assumptions are [26]:

(1) The mass flow rate, material properties, and surface temperature are constant.

(2) The contact air has constant behavior.

(3) The flow is assumed to have a uniform inlet velocity of 1 m/s and an outlet one of about 0.83m/s.

(4) All the mechanical and thermal loads are uniformly applied to the model.

(5) Fixture locations are supposed rigid, and it may cause some concentration of stress there.

For the thermal stress analysis, at first, the thermal load of the radiator and its effect is intended. The inlet and outlet temperatures are 100 °C and 40 °C, respectively. This temperature causes displacement, and when the radiator is assumed fixed from the inlet and outlet locations, stress appears. The heat transfer coefficient for the radiator combined with a fan is supposed as 70 w/m^2*K [27]. Bounded interfaces and rigid body assumptions are considered for the radiator model. The summary of boundary conditions applied to the model is shown in Table 2.

Table 2. Boundary conditions of the thermal simulation [25]

Specification	Value	Unit
Inlet temperature	100	°C
Outlet temperature	40	°C
Heat transfer coefficient	70	W/m^{2}K
Reference temperature	25	°C

The second FEA simulation is the investigation of mechanical loads on the radiator. The gravity and static loads are applied, resulting from the weight of parts connected to the radiator. The radiator is assumed fixed from the inlet and outlet locations in the static analysis. The body is assumed rigid and has bounded interfaces, and the three-dimensional triangular mesh was employed. The summary of boundary conditions is shown in Table 3.

Table 3. Boundary conditions of the mechanical simulation [25]

Specification	Value	Unit
Gravity acceleration	9.80	m/s^2
Load	10	kg-f
Reference temperature	25	°C

The next simulation is the thermo-mechanical analysis. In this analysis, in addition to the mechanical and thermal loads, the internal pressure is applied to all internal faces of the radiator, aiming to provide a closer situation to the working of the car engine. The radiator is assumed fixed from the inlet and outlet locations in the simulation. The body is assumed rigid with bounded interfaces, and the three-dimensional triangular mesh is employed. Table 4 gives the values of boundary conditions.

Table 4. Boundary conditions of thermo-mechanical simulation [25]

Specification	Value	Unit
Gravity acceleration	9.80	m/s^2
Internal pressure load	1.7	MPa
Thermal load	From the previous study	-
Weigh load	10	kg-f
Reference temperature	25	°C

Then, it is intended to study the flow analysis. The related boundary conditions are according to Table 5. It is supposed that the internal liquid is water and has permanent features.

Table 5. Boundary conditions of CFD analysis [25]

#	Specification	Value	Unit
Inlet	Heat	400	k
	specified velocity (boundary normal)	1	m/s
	Particle release (Forward)	Yes	-
Outlet	Heat	300	k
	specified velocity (boundary normal)	0.83	m/s

Finally, geometrical optimization is addressed. We aim to minimize the volume of the radiator as a target, and two dimensions of the tube (10×36) are employed to find the best solution. In the next section, the results are presented.

The radiator's primary task is to lower the working temperature in the engine so that it works as well as possible. The cooler acts here as a heat exchanger when you fill the cooler with coolant. This liquid circulates around the engine with the help of an impeller that sits in the water pump and is driven by a drive belt on the engine. This causes the liquid to be heated up by the temperature in the engine and then transported to the cooler that is normally located in the front of the car. In the cooler, the liquid flows through small channels, these channels have flanges between them and it is the flanges that help with the heat exchange. With the help of either a fan, speed wind or both, the air cools down the ducts and flanges with the hot coolant and in this way the heat is led away. The cooling fan and the cooling water pump have a power consumption that is usually around 5% of the engine power [22].

Heat transfer can take place in three different ways, two of which are convection and conduction.

Convection works so that the hot and cold particles of the liquid mix and in this case the cold particles will seek out warm and cool them down and vice versa where hot particles seek out cold particles and heat them.

The cooler's primary method of heat transfer is convection, which involves heat transfer from a solid wall to a moving fluid, which in our case is the cooling fluid that is drawn around the system with the help of a pump. In the same way, we also use convection with the help of the air that flows through the cooler and cools down the outer material of the cooler, which in turn can cool the liquid on the inside. So, while the air cools the material, it also transports away heat from the liquid, which heats the material from the inside.

In this case, the convection is said to be forced because the liquid is made to flow with the help of the pump.

Water as a coolant is not optimal as there is a risk of corrosion and deposits in the engine. Therefore, desalinated water is sometimes used. However, it is most common in engines that so-called coolant is used.

The most common type of coolant is what is commonly called glycol (ethylene glycol), but which is actually called ethanediol. Ethanediol has two tasks in the car's cooling system. The main task is to prevent frost formation in the radiator and cooling system and the secondary task is to prevent corrosion in the cooling system. Usually, the glycol is dissolved in water. The mixture consists of about 50% water and 50% glycol. This results in a freezing point of around –40 °C and a boiling point of around 108 °C. The pressure in the cooling system then raises the boiling point further, to around 120 °C.

In addition to ethylene glycol, propylene glycol is also common in cooling systems. Both are colloquially known as glycol, which makes them difficult to distinguish. It is important not to mix ethylene glycol with propylene glycol as this entails a high risk of lump formation and thus a stoppage in the cooling system. Many manufacturers color their glycols, usually red, blue or green. However, there is no industry standard for which type should have which color, which means that, for example, a blue glycol can be both ethylene glycol and propylene glycol.

Ethylene glycol is more common than propylene glycol today, as propylene glycol has somewhat poorer heat conduction and rust protection capabilities. However, ethylene glycol is very toxic. Propylene glycol is also toxic, but to a lesser extent. Drinking a small amount of ethylene glycol can be enough to cause symptoms such as unconsciousness, convulsions and severe kidney damage.

Materials: We will carry out the analysis with two different materials. Copper and an alloy with copper and aluminum. The most common material currently used in the manufacture of coolers is aluminum. But we want to compare the result with the values of a cooler made of copper as well as an alloy of these two materials to see which of the materials is preferable. Here you also have to consider the cost compared to performance in order to find out which material is the most affordable for its function. After all, the most important aspect is to be able to ensure a good function.

Temperature: Ambient temperature is set to 20 °C, which is considered the normal summer temperature in Sweden.

The temperature of the coolant is set to 120 °C, which is the liquid's boiling point, and the temperature out is set to 80 °C, which is considered reasonable.

Pressure: The pressure is set to 2.5 bar, which is overpressure for a normal cooler.

Gravity: 9.81 m/s^2.

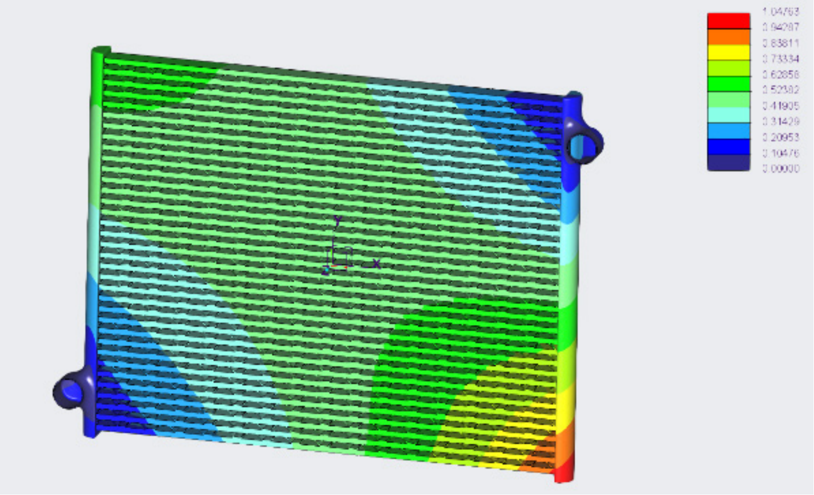
Load from above: 250 N which corresponds to approx. 25.5 kg.

Max yield stress: 220 MPa for copper and 240 MPa for the alloy.

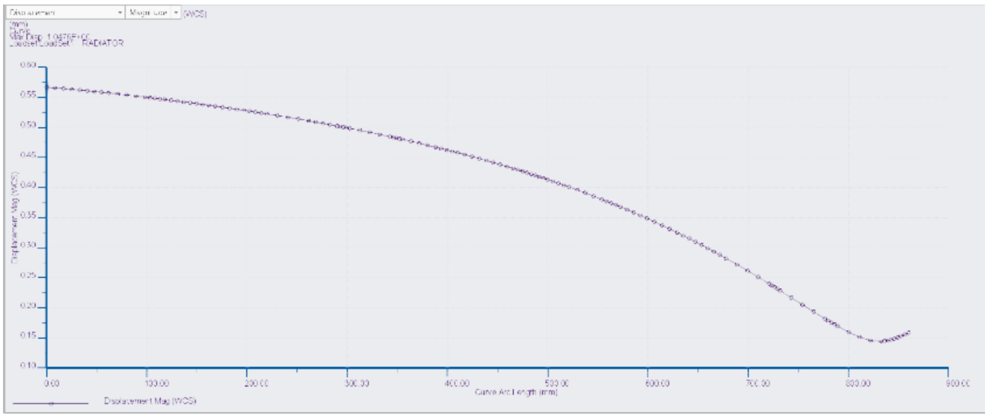
3. Results

In this section, based on the above-mentioned procedure for modeling, the results of FEA simulations are presented. For the thermal analysis, the displacement distribution is illustrated in Figure 4a. It is clear that the inlet tank side has more displacement and stress (1.04 mm) due to the more gradient of temperature and distance from the fixture location. Figure 4b shows the detailed displacement distribution of the upper tube of the radiator along the Y-Axis from the inlet to the outlet. It is obvious that the minimum displacement occurs when the flow is moving out of the radiator, which is around 0.15 mm. Figure 5 shows the Tresca theory's failure index. The maximum value is about 4.62, near the inlet and outlet, as expected. It results from the boundary condition simplification assumption.

For the mechanical analysis, the resulting displacement distribution is represented in Figure 6a. There is a symmetric distribution because of the symmetric geometry and loads, although it includes some simplification of the load distribution. Obviously, the maximum displacements and stresses (0.02 mm) occur far from the fixtures' location. The detailed displacement distribution of the upper tube of the radiator along the Y-Axis from the inlet to the outlet is shown in Figure 6b. Displacement has risen from the inlet to the outlet in the upper face of the tube. Figure 7 shows the Tresca theory's failure index. The maximum value is around 0.018, near the inlet and outlet as expected. It ensues from the boundary condition simplification assumption.

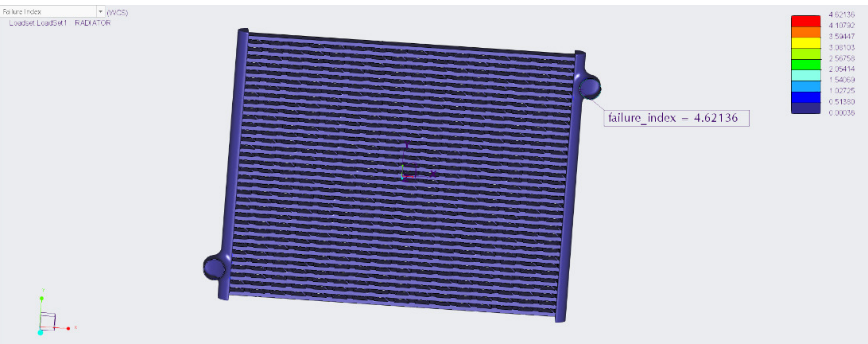


(a)



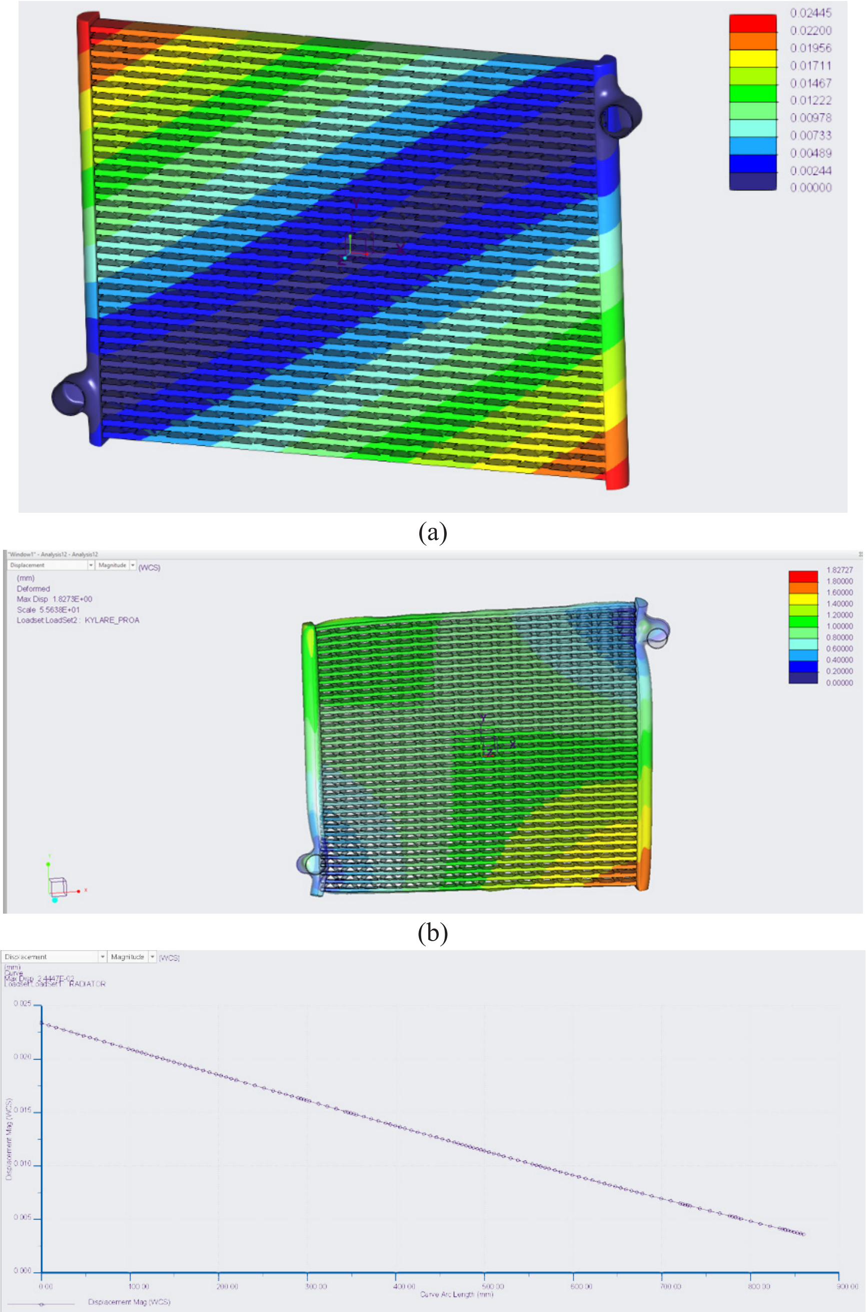
(b)

Figure 4. Resulting displacement distribution of the thermal analysis: (a) displacement distribution, (b) displacement distribution through the Y-axis1



Failure Index (WCS)
Loadset:LoadSet1 : RADIATOR

Figure 5. Failure index distribution of the Tresca theory for the thermal analysis



(a)

"Window1" - Analysis12 - Analysis12
Displacement Magnitude (WCS)
(mm)
Deformed
Max Disp 1.8273E+00
Scale 5.5638E+01
Loadset:LoadSet2 : KYLARE_PROA

(b)

Displacement Magnitude (WCS)
(mm)
Curve
Max Disp 2.4447E-02
Loadset:LoadSet1 : RADIATOR

(c)

Figure 6. Resulting displacement distribution of the mechanical analysis: (a-b) displacement distribution, (c) displacement distribution through the Y-axis

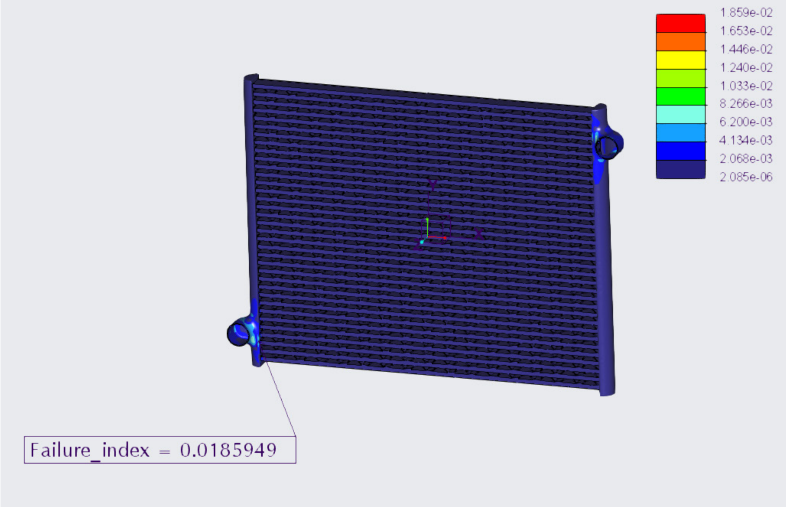
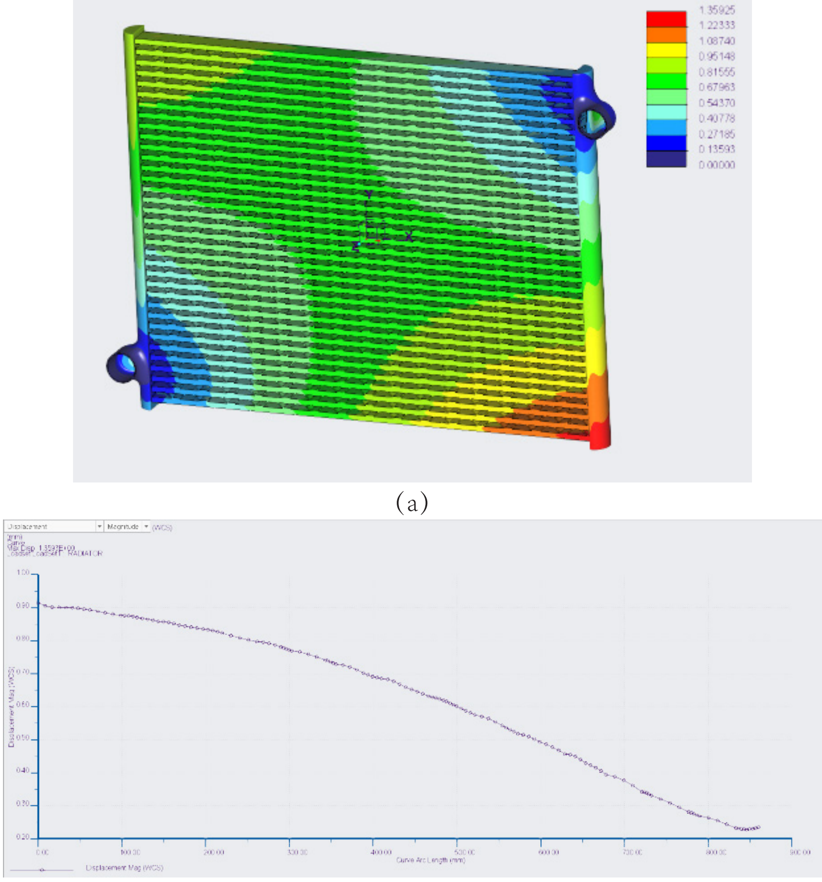


Figure7. Failure index distribution of the Tresca theory for the mechanical analysis

For the thermo-mechanical analysis, the displacement distribution is depicted in Figure 8a. There is a kind of symmetric distribution because of the symmetric geometry and mechanical loads, although thermal load causes the concentration of the stress in the inlet tank side far from the fixture location. The maximum displacement is about 1.35 mm. Figure 8b shows the detailed displacement distribution of the upper tube of the radiator along the Y-Axis from the inlet to the outlet and again from the inlet to the outlet. The displacement has risen through the upper face of the tube. Figure 9 shows the Tresca theory's failure index. The maximum value is around 4.58 happened near the inlet side as expected.



(a)

(b)

Figure 8. Resulting displacement distribution of the thermo-mechanical analysis: (a) displacement distribution, (b) displacement distribution through the Y-axis

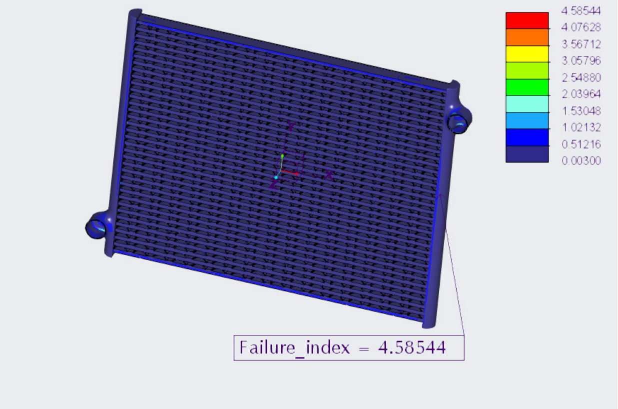


Figure 9. Failure index distribution of the Tresca theory for the thermo-mechanical analysis

Thermal stress causes seven times more displacement than a mechanical one. Although when they are combined, the value reaches 1.5 mm. It is seen that the radiator is pretty safe because in most locations, there is not any failure index of more than 1. It is clearly indicated in Figure 4a and Figure 8a that both the pure thermal and thermo-mechanical simulations have similar displacement distribution behavior. This indicates the priority of the thermal stress against the mechanical loads in the failure of the radiator. Hence, the thermal load has an important role in the failure, and it must be considered in the radiator designing process.

The results of the flow analysis are shown in Figures 10-13. Particle release starts from the inlet, and they have a forward direction into the radiator. Figure 10 illustrates velocity along X-axis. Obviously, some tubes in the middle of the radiator have lower values than others. Also, the inlet and outlet both have a medium value of velocity. In Figure 11, the internal pressure of the radiator is shown. The pressure is more in the inlet than in the outlet because they have different temperatures and conditions. Enthalpy represents the pretty harmonic distribution, as shown in Figure 12.

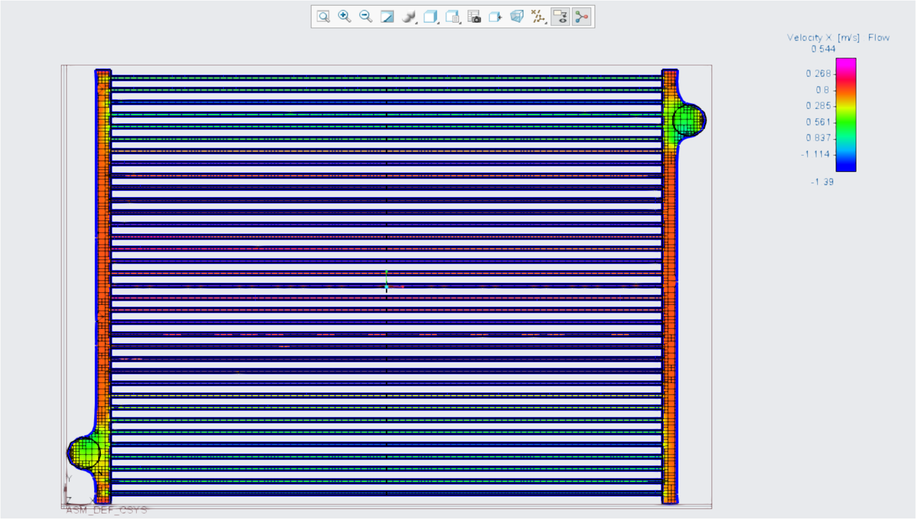


Figure 10. X-Velocity

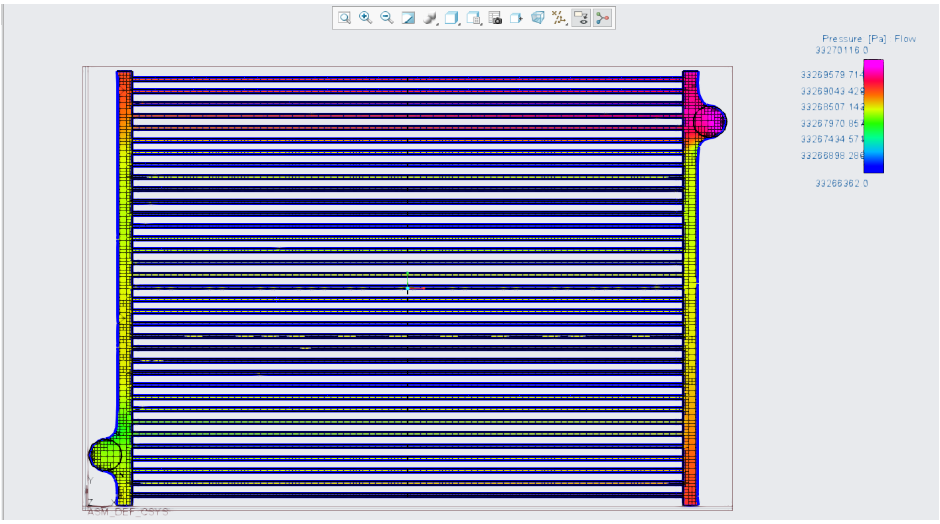


Figure 11. Pressure

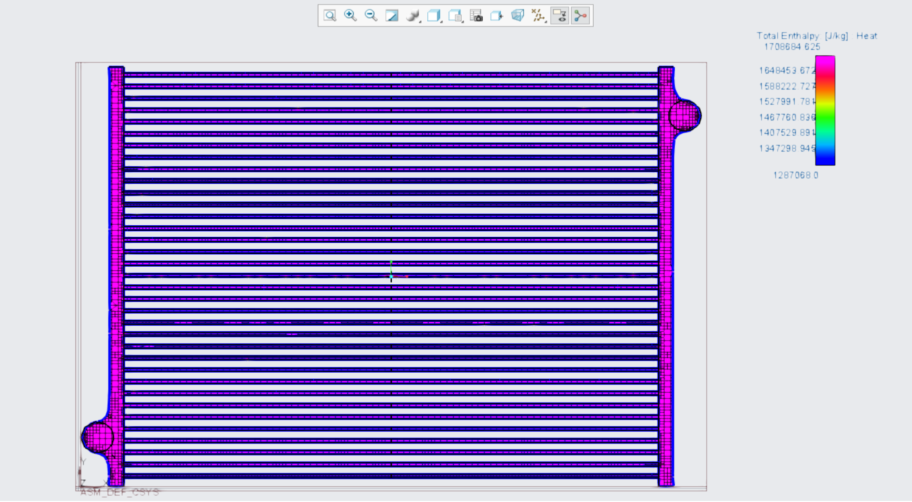


Figure 12. Total Enthalpy

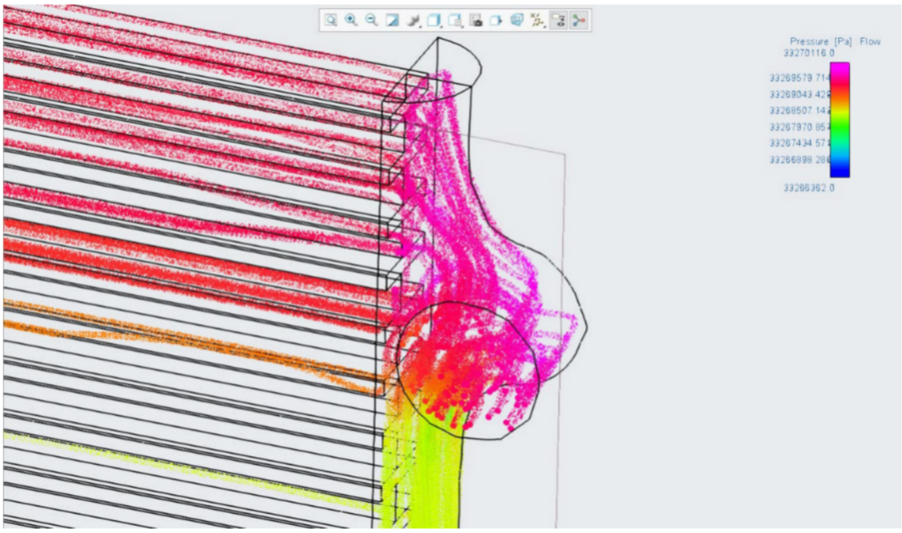


Figure 13. The flow analysis in connection to the pressure flow

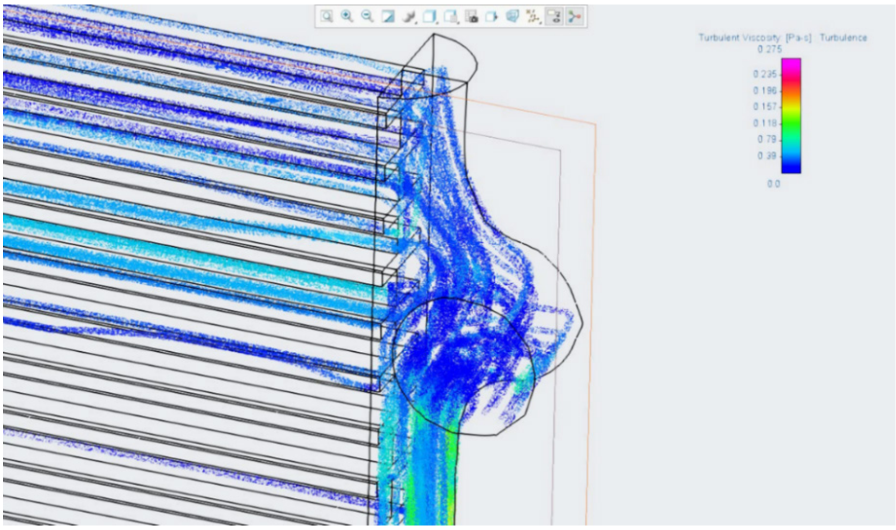


Figure 14. The flow analysis in connection to the turbulent viscosity

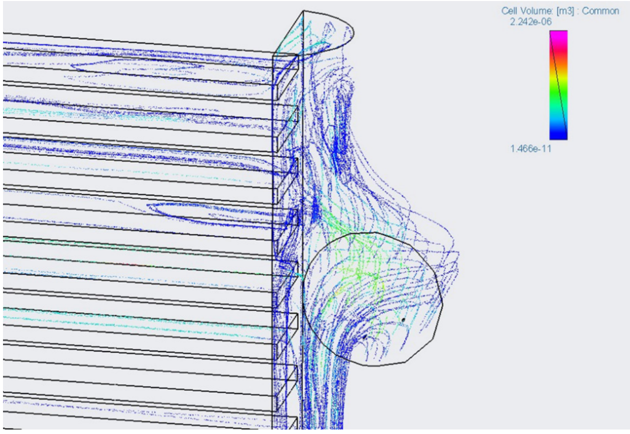


Figure 15. The flow analysis in connection to the cell volume

4. Discussion

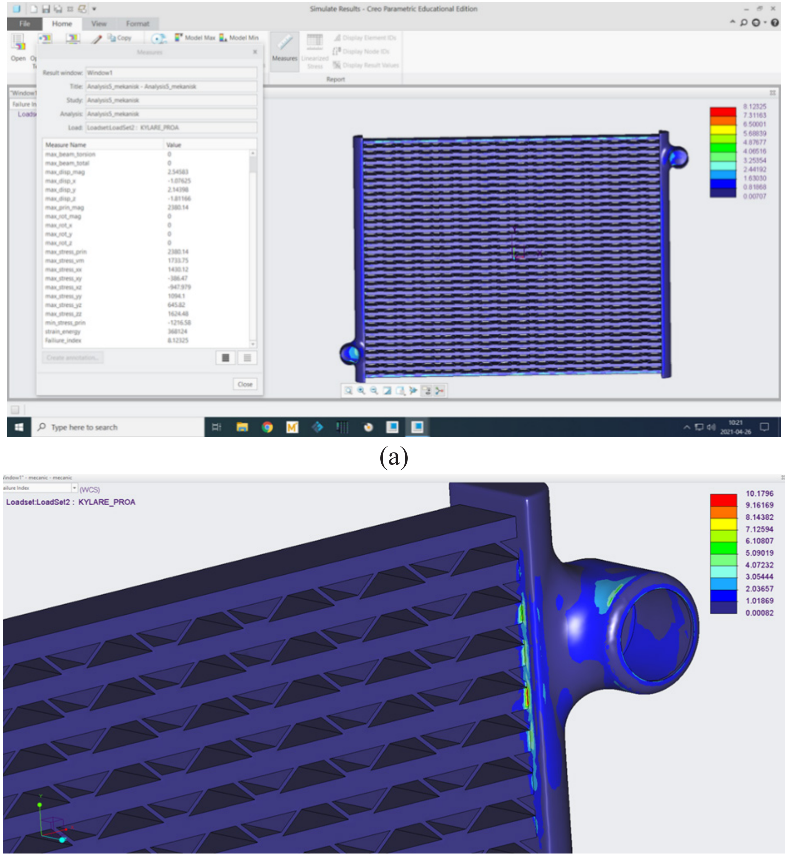
The results show that thermal stress causes 7 times more displacement than a mechanical one. Although when they have been mixed together the value grows to 1.5 mm. It is shown that the radiator is pretty safe. Because in most locations there is not any failure index of more than 1. Surprisingly results are about the effect of thermal stress and load on radiator mechanical failure.

Both pure thermal and thermo-mechanical simulation have similar displacement distribution behavior and it shows the priority of the thermal stress against mechanical loads in the failure of the radiator. So thermal load has an important role in failure and it must consider in the radiator designing process.

After carrying out the mechanical analysis (Figure 16) with regard to the failure index, it can be seen that the cooler will not make it because at certain points you see that the index is greater than 1, which corresponds to failure. What you can see is that the joints at the inlet and outlet will be extremely exposed and this is where the cooler will break.

Here we have a picture of the same analysis as the picture before, but now we have changed the material to pure copper. We see no major differences in the analysis apart from achieving a higher failure index, but the cooler will still break at the inlet and outlet respectively.

Furthermore, we also looked at displacement in mm that occurs with various loads that affect the cooler. This is by fixing an outlet which will be joined with the rest of the system in the car. Of course, the cooler will also have support in general, but these points are not included in the analysis.



Result window: Window1
Title: Analysis5_mekanisk - Analysis5_mekanisk
Study: Analysis5_mekanisk
Analysis: Analysis5_mekanisk
Load: Loadset:LoadSet2 : KYLARE_PROA

Measure Name	Value
max_beam_torsion	0
max_beam_total	0
max_disp_mag	2.54583
max_disp_x	-1.07625
max_disp_y	2.14398
max_disp_z	-1.81166
max_prin_mag	2380.14
max_rot_mag	0
max_rot_x	0
max_rot_y	0
max_rot_z	0
max_stress_prin	2380.14
max_stress_vm	1733.75
max_stress_xx	1430.12
max_stress_xy	-386.47
max_stress_xz	-947.979
max_stress_yy	1094.1
max_stress_yz	645.82
max_stress_zz	1624.48
min_stress_prin	-1216.58
strain_energy	368124
Failure_index	8.12325

(a)

(b)

Figure 16. Mechanical analysis

In this analysis, the heat spread in the cooler is shown when the inlet temperature is 100 degrees and the outlet temperature is 80 degrees. What we readout is that it is the middle part of the cooler that is responsible for the cooling and not the edges. According to the results we got, the middle part of the radiator is 25 degrees, and this is because we could not get any liquid into the radiator. With liquid, it would have looked different as the heat in the radiator had decreased from 100 degrees down to 80 degrees instead of going from 100 to 25 and then to 80 which is not reasonable.

This analysis (Figure 17) shows how the heat affects the cooler together with the mechanical aspects. The top image shows that the shear appears to decrease when pooling the analyses. Now we only see a slightly larger impact in the lower right corner mine to displacement. Furthermore, we see that the inlet will achieve higher stress when it comes to the wrong index and ends up a bit above one, which means that the material will not hold based on the loads we put on the analysis.

4.1 Optimization of Volume

To see if an improvement to the cooler could be made, the volume of the cooler was examined. A larger volume means that a larger amount of coolant can flow through the cooling system and thus better cooling is achieved. The optimization of the volume was done with PTC Creo's analysis tool. The original volume was measured as:

$$V\ 1 = 7747209\ \text{mm}^3 = 7.747\ \text{liters} = 0.0077\ \text{m}^3$$

After that, parameters were specified that could not be changed. Since the outer dimensions of the cooler are critical for the fit in the engine, these were specified as locked, that is, the total height and width of the cooler could not be changed. Parameters that were allowed to change were the height and width of the flanges. In the original model, the flanges were 40 mm × 10 mm (Figure 18). They were allowed to vary between 36 mm and 44 mm for the width and 9 mm and 11 mm for the height, respectively, to find the optimal volume.

The optimization resulted in the dimensions of the flanges changing from 40 mm to 44 mm in width and from 10 mm to 11 mm in height. The outer dimensions were, as previously described, locked and thus not changed. The optimization is presented in a graph where you can see how the volume is increased. The new volume was measured to:

$$V\ 2 = 8292732\ \text{mm}^3 = 8.2927\ \text{liters} = 0.0083\ \text{m}^3$$

This means that the volume increased by $\Delta V = 545523$ mm^3 = 0.546 liters.

From the mechanical analysis carried out, we have found that the cooler's weakest link is at the inlet and outlet, where you will get high pressure. It is in these two points that deformation will mainly take place. Both at the pressure that has been applied and also at the displacement that may occur from the loads that will load the cooler.

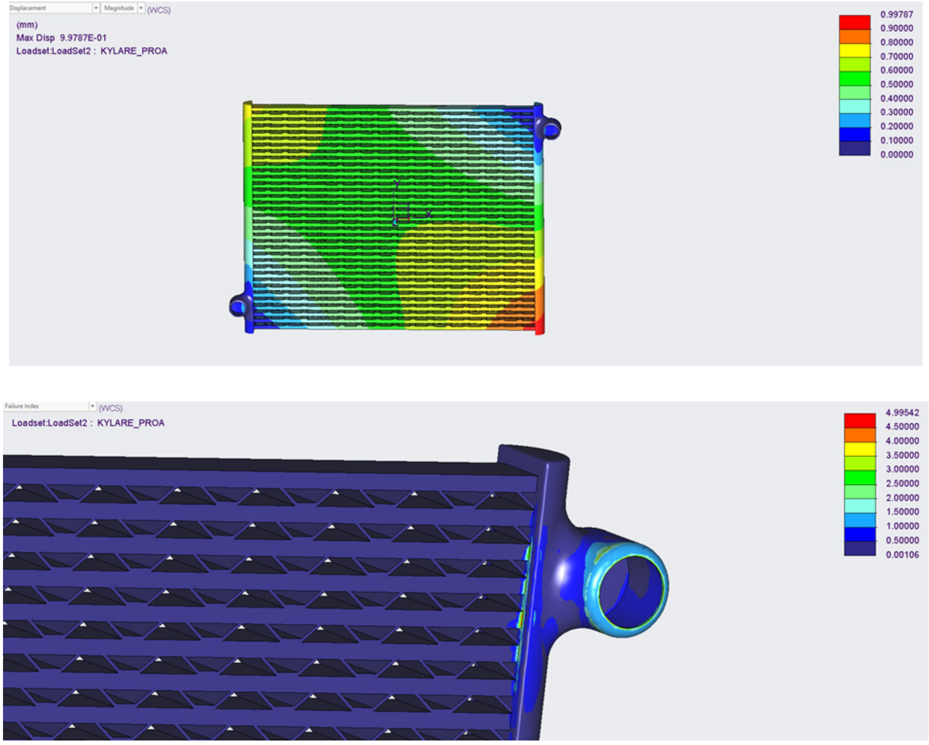
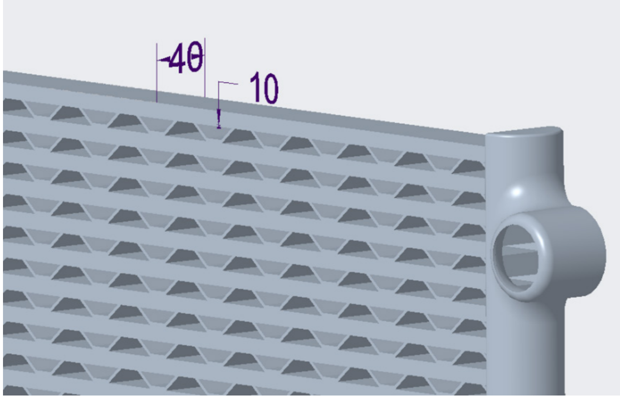
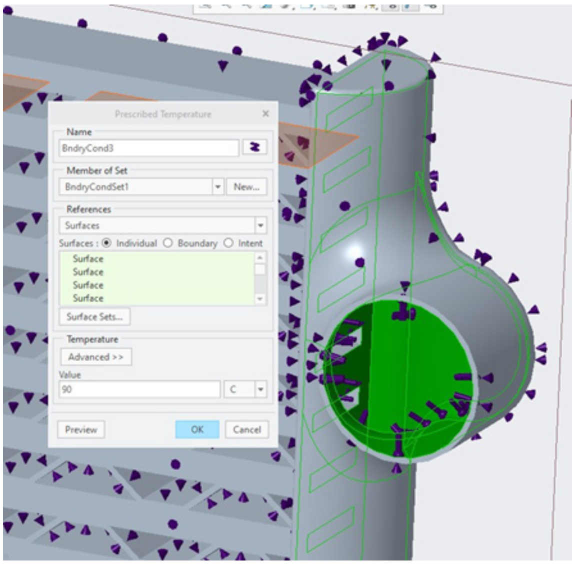


Figure 17. Thermo-mechanical analysis




Prescribed Temperature

Name: BndryCond3

Member of Set: BndryCondSet1

References: Surfaces

Surfaces : ● Individual ○ Boundary ○ Intent

Surface
Surface
Surface
Surface

Temperature

Value: 90 C

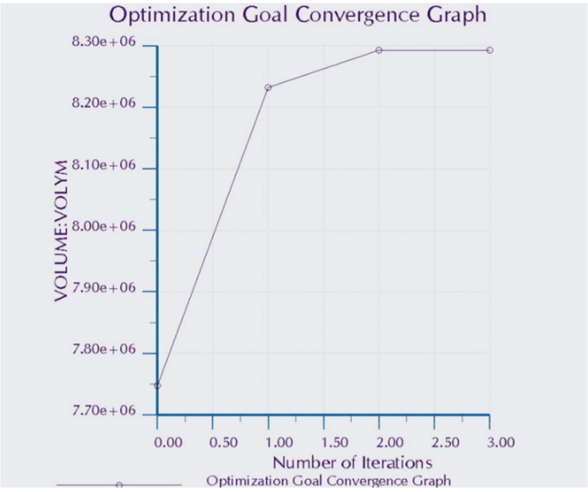


Figure 18. Graph for optimization of volume

Based on the heat analysis, the cooler will spread the heat over the entire cooler. The absolute largest part of the cooling comes from the cooling fins, as they do not have direct contact with the liquid but only cool the material around the liquid. It also appears that there is a good distribution of the heat spread in the cooler as no part stands out more than another.

The optimization resulted in the volume increasing by $\Delta V = 545523\ mm^3 = 0.546$ liters.

As the volume increases, it means that we should get better cooling as more coolant can flow through the radiator. The theoretical amount of heat (the cooling effect in this case) is calculated as $Q=m'*c*\Delta T$ and the relationship between the mass flow and the volume flow is obtained from $m'=\varrho * V'$ (Alvarez, 2006). With these two equations, we can easily see that a larger volume gives a higher volume flow, which in turn gives a larger mass flow. The greater mass flow then results in a higher theoretical cooling effect. In order to check that this is correct and how much better the cooling effect really is, a flow analysis is required, but as previously mentioned, we cannot carry that out with the student version of PTC Creo.

What we see in the heat analysis is that everything works as it should, which means that the cooler will work as it should from that point of view. We also see that it is at the heat sinks that it cools the most. If you want to optimize the cooler, you can make more but smaller cooling channels through the cooler, which all have cooling fins connected. In analysis 3, we get an unreasonable result when the heat in the cooler goes from 100 degrees to 25 and then up to 85. We got this result when the cooler only had one heat in the inlet and one heat in the outlet, this meant that the middle part of the cooler was not heated up as it should have done if you had had coolant in. The result we should have gotten would have looked like the temperature decreased exponentially as the temperature had decreased a lot at the beginning and then less at the end, until it reached the temperature.

The radiator will be deformed around the outlet and inlet because in the analysis we have fixed these points and the remaining part of the radiator which will then expose these points to the accumulated load on the radiator.

In the thermal analysis, one has gained an idea of how the radiator works regarding the cooling of the coolant when it is used during the cooling process itself. First, the liquid is hot at around 90 degrees Celsius when it flows into the inlet and then it is cooled down inside the radiator due to the fan and then it is at its coldest. When the process is finished, the radiator itself becomes warm again, but not the at same temperature as at the start.

When it comes to thermo-mechanical analysis, it's about all the safety errors within the system. Therefore, points are shown where the design was unstable and hence had more chance of being destroyed. What you see in the analysis itself is the point where there is the most chance of being destroyed when it comes to the inlet pipe where there are the most pressure & temperature increases i.e.

thermodynamic loads.

In the displacement analysis, it is the deformations that are shown that the thermodynamic loads could subject the radiator too. The maximum deformation is found at the right-bottom edge of the inlet pipe. This is obvious as it is a liquid that comes with strong pressure and high temperature which leads to the radiator's edge being deformed.

One proposed solution was to produce a gradual change when it comes to reducing the thermo-mechanical loads when discussing the inlet. This solution proposal has been tackled and managed to go through, which has led to an increase in security errors at the entrance and that is exactly what we wanted to arrive at.

The last proposed solution is the actual optimization of the flow in the radiator, where the volume is minimized by 3,962.51 square centimeters and 86 centimeters from axis to axis, in other words, the width of the radiator was shortened by 4 mm, which led to a reduction in volume. This leads to, among other things, more space for vehicles for other components. A large radiator is not always good as it is the effect itself that proves whether it is good or not.

4.2 Limitation and Future Work

There are restrictions on how Creo Simulate can establish its mathematical limit, such as restrictions on how to apply loads or constraints on merged surfaces. For instance, if the user specifies the von Mises stress value as an optimization limit (for instance, 20,000 si), Creo Simulate will seek to discover a more effective model that still satisfies both that restriction and any goal you specify by moving the model toward 20,000 psi.

The fact that this study focused more on the useful aspects of the thermal performance of the car radiator might be used to prevent the car radiator from overheating. However, due to the limited space in vehicles, relocating the heat exchanger or employing new materials may be advantageous in the development of improved heat exchangers.

4.3 Future Work

The prospective extensions of this work could be as follows:

- More simulations with transient driving cycles, improved models, and input information tailored to the individual components. This would be an appropriate technique to increase knowledge and confirm the chosen arrangement. It is also advised to do practical testing to gather pertinent information on radiators, fans, and the active cooling system.
- Additional arrangements and options might also be looked into. Condenser placement is one example.
- The incorporation of an AC system, which is not detailed in this paper, could result in even more energy and weight savings. In this situation, adding a chiller loop to enable heat pump mode ought to be looked into.
- Additionally, it is advised to use extra heat for cabin heating as necessary. Removing the LTC heater might be an option since the cabin also requires an auxiliary heater.

5. Conclusions

This study successfully performed a coupling analysis between the thermal and mechanical conditions in a typical Truck radiator. Distribution of the displacement and the failure index of Tresca theory was provided for three cases. Then, flow analyses of the radiator were investigated, including the variations of velocity, pressure, and enthalpy. The results indicate the high importance of the thermal loads necessitating their special consideration in the radiator designing process. This paper also shows the ability of PTC-Creo in the effective simulation of mechanical and thermal conditions in the radiator of a heavy-duty cooling system, which can be deemed as a reliable and economical tool for industrial applications.

When it comes to thermal analysis, it has been visualized and understood how a radiator works when it comes to cooling down the coolant in the radiator for reuse in the cooling process. The hot coolant enters the inlet where it is the hottest at 90 degrees Celsius in temperature and is slowly passed through the radiator where the heat is removed in the middle with the help of wind and a fan and there the radiator should be the coldest as shown in the analysis.

Regarding the thermo-mechanical analysis, it showed the failure safety of the whole system. It, therefore, showed which points were weakest in the design and which had the greatest risk of failure. What can be seen from the analysis is the point that has the greatest risk of failing the inlet pipe, which is understandable as it is the point that is exposed to the most thermodynamic loads such as pressure and high temperatures. The next analysis was the displacement analysis, it shows the deformation the thermodynamic loads could subject the radiator to and the maximum deformation is in the lower right corner at the inlet pipe. This is also understandable as the liquid enters with a high pressure and also a high temperature which causes the plate to deform.

In the solution proposals (Figure 19), the first thing was to make the entire plate thinner. This was done in

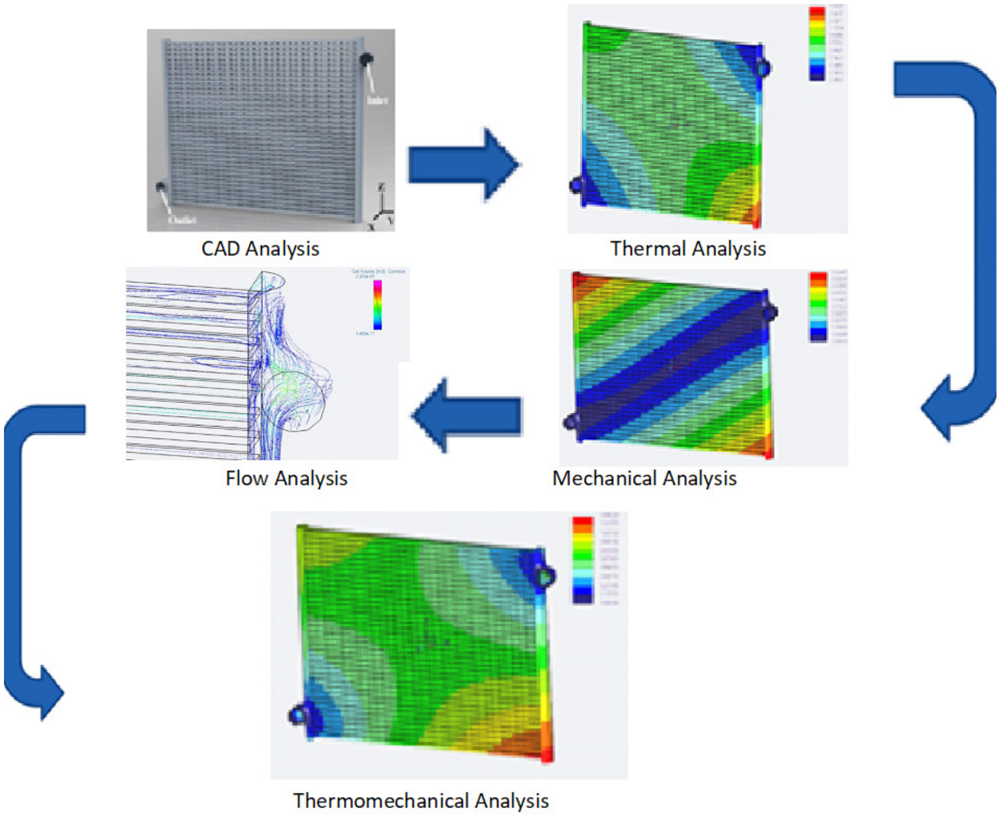


Figure 19. Graph for optimization of volume

the hope that it will be cheaper to manufacture while not too much of the strength is lost in the process. What the results of the analysis say is that it was the case, in the error safety analysis it can be seen that the error safety is not compromised too much over the whole system, there are also places like at the beginning where the error safety benefits from this improvement as it is spread out more and got better values. The displacement analysis also indicates that a thinner plate is not a bad idea as the maximum deformation of the system with the thin plate was less than the one with the thicker plate. The next solution was to create a gradual transition to reduce the thermo-mechanical loads on the entrance. The group has succeeded in this, this transition has, according to the results of the analysis, succeeded in increasing the total failure safety on the inlet pipe, which was what was the goal.

Conflict of Interest

There is no conflict of interest.

References

[1] Kasu, P.R., Ganesh, N.B., Kumar, K.S., et al., 2017. Analysis of heat dissipation in radiator of SI engine. International Research Journal of Engineering and Technology. 4, 160-166.

[2] Rahmati, A.R., Gheibi, A., 2020. Experimental and numerical analysis of a modified hot water radiator with improved performance. International Journal of Thermal Sciences. 149, 106-175.

[3] Amrutkar, P.S., Patil, S.R., 2013. Automotive radiator performance-Review. International Journal of Engineering and Advanced Technology. 2(3), 563-565.

[4] Eitel, J., Woerner, G.T., Horoho, S., et al., 1999. The aluminum radiator for heavy duty trucks. SAE Technical Paper. 1, 3727. DOI: https://doi.org/10.4271/1999-01-3721.

[5] Aravindkumar, N., Vignesh, S., Kumaran, P., et al., 2020. Computational and numerical analysis of radiator with different tube structures and nano fluid as coolant. Materials Today: Proceedings. 45(Special Issue). DOI:10.1016/j.matpr.2020.07.605.

[6] Sheikhzadeh, G., Hajilou, M., Jafarian, H., 2014. Analysis of thermal performance of a car radiator employing nanofluid. International Journal of Mechanical Engineering and Applications. 2(4), 47-51.

[7] Molana, M., 2017. On the nanofluids application in the automotive radiator to reach the enhanced thermal performance: a review. Journal of Heat and Mass Transfer. 4(4), 168-187.

[8] Abbas, F., Ali, H.M., Shah, T.R., et al., 2019. Nanofluid: Potential evaluation in automotive radiator. Journal of Molecular Liquids. 297, 112014.

[9] Sathyan, R., 2016. Analysis of automobile radiator using computational fluid dynamics. International Journal of Latest Technology in Engineering, Management & Applied Science (IJLTEMAS). 5, 6.

[10] Chidley, A., Roger, F., Traidia, A., 2011. A Thermal-Shock Fatigue Design of Automotive Heat Exchangers. International Journal of Mechanical and Mechatronics Engineering. 5(1), 263-267.

[11] Roger, F., Chidley, A., 2013. Thermo-mechanical fatigue design of automotive heat exchangers. European Journal of Computational Mechanics. 22(2-4), 228-235.

[12] Gu, J., Yang, X.F., Ma, T., et al., 2013. The study of automotive aluminum alloy radiator tank's leakage. Applied Mechanics and Materials. 364, 134-137.

[13] Baou, M., Afsharpanah, F., Delavar, M.A., 2020. Numerical study of enhancing vehicle radiator performance using different porous fin configurations and materials. Heat Transfer-Asian Research. 49(1), 502-518.

[14] Kim, T.K., Lee, S.M., Pae, S.M., 2017. Method of predicting radiator temperature distributions for thermal fatigue analysis. Journal of Mechanical Science and Technology. 31(10), 5059-5066.

[15] Amrutkar, P.S., Patil, S.R., Shilwant, S.C., 2013. Automotive radiator-design and experimental validation. Research and Development (IJAuERD). 3(4), 1-10.

[16] Shariff, K.B., Abdullahi, B., Abubakar, S.B., 2018. Modelling and simulation of car radiator: Effects of fins under the atmospheric condition of Kano, Nigeria. Journal of Advanced Research in Fluid Mechanics and Thermal Sciences. 48(1), 1-16.

[17] Mia, S., Hossen, S., 2019. Numerical observation of heat transfer and pressure behaviour in an automobile radiator. Journal of Mechanical Engineering. 49(2).

[18] Patel, H.V., Subhedar, D.G., Ramani, B., 2017. Numerical investigation of performance for car radiator oval tube. Materials Today: Proceedings. 4(9), 9384-9389.

[19] Kasu, P.R., Ganesh, N.B., Kumar, K.S., et al., 2017. Analysis of heat dissipation in radiator of SI engine. International Research Journal of Engineering and Technology. 4, 160-166.

[20] Bonato, M., 2020. From Manufacturing Process Control to In-Field Reliability: Case Study of Automotive Heat Exchangers [Internet]. 2020 Annual Reliability and Maintainability Symposium (RAMS) [Published 2020 January 27]. pp. 1-7. Available from: https://dl.acm.org/doi/10.1109/RAMS48030.2020.9153724.

[21] Goyal, N., Kaushik, A., Ram, M., 2016. Automotive water cooling system analysis subject to time dependence and failure issues. International Journal of Manufacturing, Materials, and Mechanical Engineering (IJMMME). 6(2), 1-22.

[22] Henrik, A., 2006. Energiteknik Edition. pp. 360-370. ISBN: 9789144045092.

[23] Tenkel, F.G., 1974. Computer Simulation of Automotive Cooling Systems. SAE Technical Paper. 2, 1. DOI: https://doi.org/10.4271/740087.

[24] Global Auto Parts & Accessories Marketplace [Internet]. SCANIA Radiator 1365371. Available from: http://bonamotor.en.gasgoo.com/auto-products/1450301.html.

[25] Aluminium Matter Organization UK. [Interner]. Available from: https://european-aluminium.eu/.

[26] Ellis, D.E., 1960. Mechanical properties of aluminum alloys at various temperatures. Atomics International. Div. of North American Aviation, Inc., Canoga Park. United States: N. Available from: https://www.osti.gov/biblio/4097760.

[27] Peyghambarzadeh, S.M., Hashemabadi, S.H., Naraki, M., et al., 2013. Experimental study of overall heat transfer coefficient in the application of dilute nano fluids in the car radiator. Applied Thermal Engineering. 52(1), 8-16.