



Connecting Rod Fatigue Life Analysis Using Ansys Workbench 2021 R1

University of Wales Trinity Saint David's
School of Engineering
Module: ACHA6010

Contents

1	Introduction.....	2
2	Methodology	3
2.1	Validation setup	3
2.2	Model Design.....	4
2.3	Model assembly and setup within Ansys	5
3	Test Results.....	10
4	Results Analysis	11
4.1	Validation Analysis.....	11
4.2	Conrod study Analysis	11
5	Conclusion.....	13
6	Bibliography.....	14
7	Appendices	16

1 Introduction.

Engineering software such as Solidworks, Ansys and Recardo wave to name a few are a large part of engineering design today. This report will evaluate the effectiveness of these engineering tools by testing an engine assembly in Ansys Workbench. Through this we will be able to determine where the limitations of the software are. From a practical aspect.

The engine assembly that has been tested consists of the complete crank assembly. The crank assembly is a to scale model of a Yamaha SZR 660 crank excluding the con-rod and the piston which are aftermarket racing components. The con-rod is a H-beam racing con-rod from CP-Carrillo. And the piston is a 11:1 compression 2mm overbore 102mm CP-Carrillo. The weight of the components has also been measured to ensure that the mass of the components in Ansys are representative of the real components or assemblies. This is important as the mass of the components are what control the loads generated when the assembly is rotating.

Ansys will be used to examine the loadings from the little end of the connecting rod at a few RPM ranges. This is to estimate the limitations of the con-rod, or its expected fatigue life if pushed beyond the existing rev limit. Also, this analysis of a 'performance' part may bring some clarity to the design considerations that took place.

The final section of the report will discuss the limitations with Ansys and such alike software. This may include its accuracy, useability and limitations due to its inability to simulate dynamic simulations. As well as potentially needing a server to run more complex simulations.

2 Methodology

2.1 Validation setup

It is important to first validate the Accuracy of Ansys when simulating a structural and thermal model. This was done using a bimetallic strip verification example (Dubal Software, 2018). This compares a calculated theoretical result to a simulated and calculated theoretical result. Thus, validating the theoretical accuracy of the software.

The bimetallic strip consists of copper and invar strips of 100mm long 5mm wide and 1mm thick bonded together. The model for this was built in Solidworks and moved into workbench as a static structural model. The properties for the materials are supplied on the verification example, so these are used to specify the material properties within the engineering data tab in Ansys workbench. With that done the materials were applied to the model and the thermal load of +100 degrees centigrade also applied as done for the verification example.

With the bimetallic strip we are looking at the deformation due to the difference in expansion due to temperature change. Therefore, this is the solution output set in Ansys. The solution was run multiple times at different mesh densities. These were 4,2,1,0.5,0.25 and 0.125mm mesh densities. This is to check the influence mesh density has on the accuracy of the results.

2.2 Model Design

The model is designed from a Yamaha SZR660 bottom end. This consists of the two crank webs, crank pin, big end needle bearing, big end shims, connecting rod, gudgeon pin, and piston. The crank webs, crank pin, big end bearing, con rod and gudgeon pin are all modelled to the dimensions of the real parts (**see Figure 1/2**). The piston is a rough approximation of the real part, but it will be set up to have the same mass as the real part. The complete model assembly can be seen in **figure 2**.



Figure 1 – Real Crank assembly and Carrillo Rod



Figure 2 – Modelled Crank assembly

2.3 Model assembly and setup within Ansys

The assembly in Ansys mostly consists of revolute joints due to most components needing to rotate (**Figure 3**). The largest group of connections are due to the caged needle bearing at the big end of the con-rod, consisting of 22 needles and a cage.

Under the 'bottom end' tab we have more revolute joints. One for the crank bearing within the big end bearing cage. Another for the big end of the con-rod around the big end bearing cage. And two more for the two big end shims either side of the con-rod around the crank pin.

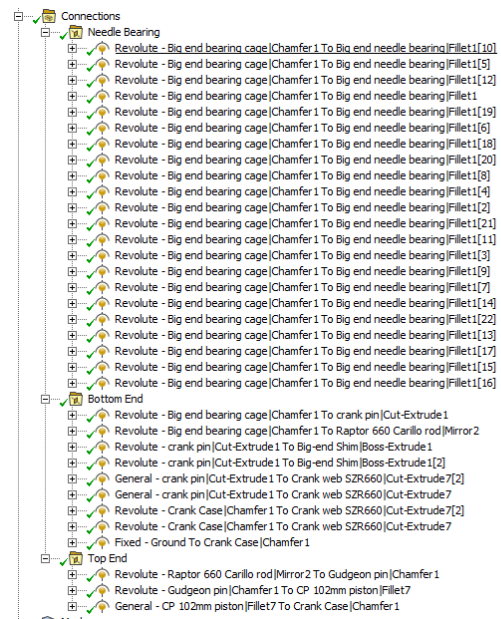


Figure 3 – Connections in Ansys

The two general joints are fixing the crank pin to the two crank webs. Replicating the press fit that exists there. And the revolute joints below that mount the crank shaft in the crank case. Replicating the main bearings.

For the top end we have two more revolute joints for the little end to the gudgeon pin. The other for the gudgeon pin to the piston. The last general joint is a for the piston in the bore of the crank case model. being allowed to only move up and down on that axis. This completes the assembly process.

The first task for the setup is to add the engineering data required for the study. Since we are studying the connecting rod then that is the component that must have the correct material properties. Unfortunately, CP-Carrillo do not disclose the exact material used for their connecting rod. Must be a trade secret. Despite this we can find material properties for a commonly used steel for performance forged con rods. This being the 4340 chromoly steel (Creason, 2013). The properties for this steel were found at (Matweb). In **figure 4** can be seen the material properties input for the 4340 chromoly steel.

The piston was also assigned the correct material properties. The exact alloy CP-Carrillo was found in the Q&A section on their website (CP-Carrillo). The material they use for their pistons is 2618-T61 aluminium. The properties for the 2618-T61 alloy were found at (Matweb).

7	STEEL 4340	Exp	Engng. Frac. Mech. Vol 21. No. 1. pp 31-48. 1985 Johnson + Cook
8	Structural Steel	Ger	Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1
*	Click here to add a new material		

Properties of Outline Row 7: STEEL 4340				
	A	B	C	D E
1	Property	Value	Unit	
2	Material Field Variables	Table		
3	Density	7.85	g cm ⁻³	
4	Isotropic Elasticity			
10	Tensile Yield Strength	862	MPa	
11	Tensile Ultimate Strength	1282	MPa	
12	Isotropic Thermal Conductivity	44.5	W m ⁻¹ K ⁻¹	
13	Specific Heat Constant Pressure, C _p	0.475	J g ⁻¹ K ⁻¹	
14	Johnson Cook Strength			
23	Bulk Modulus	159	GPa	
24	Shear Modulus	78	GPa	

Figure 4 – 4340 steel properties.

Other components were assigned materials from the engineering data library that closely fit their material. Their exact material is not important unless that component is being studied.

The next step was to link the engineering data to the engineering data in the rigid dynamics. Now the materials can be applied to all of the components in the assembly. At this point the rigid dynamics study is ready to be set up. As this CP-Carrillo Con-rod is deemed to be a performance part then in theory it has been designed to withstand a more extreme environment than the standard part. The original Yamaha SZR660 engine was designed for 48HP (Motorcyclespecs, 1997). The Carrillo rod was allegedly designed for 95HP and to last a season of racing (Veldhuijzen, 2015). And likely pushing a lot more than the standard 7250RPM rev limit.

So, for this reason the Con-rod loads were simulated at various RPM levels above the normal rev limit. These RPM levels were 8000, 9000 and 10,000RPM. These tests should indicate where the limit for this con-rod is. As well as the fatigue life if it were subjected to these higher levels of stress.

So the setup for the 8000 RPM ridged dynamics study required setting up a rotational velocity and applying it to the crank shaft. Next in the analysis settings the timings for the study required setting.

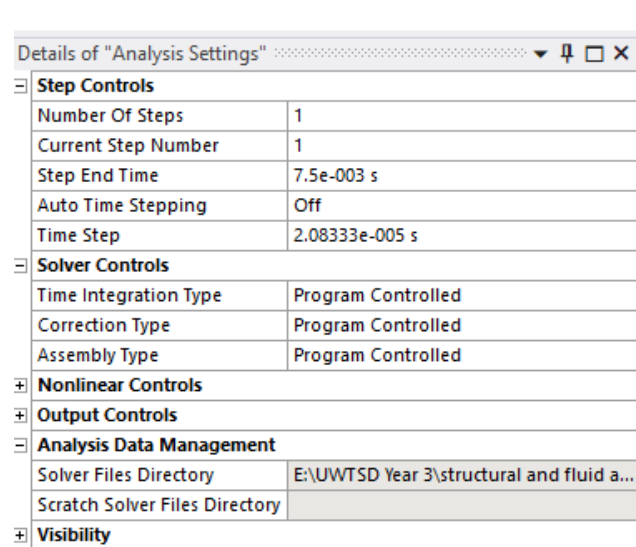


Figure 5 – Analysis settings for rigid dynamics

In **Figure 5** can be seen the settings for the 8000RPM study. To get accurate results of the peak loads, it is ideal to make the Ansys solve for one rotation and take a measurement per degree. See equations below.

To get 1 rotation in seconds:

$$1 / (8000/60) = 0.0075 \text{ s}$$

To get 1 degree in seconds:

$$1 / ((8000/60)360) = 2.0833\text{E-}5 \text{ s}$$

With that the joint between the little end of the rod and the gudgeon pin could be dragged into the solution tab and the rigid dynamics can be solved. Once solved the peak forces which were at TDC (top dead centre) can be retrieved and saved for the static structural setup. This project was then saved, and saved as two other projects, for the 9000 and 10,000 RPM studies. For those studies the RPMs were changed to their respective ranges and the analysis settings corrected with the right timings as done for the 8000RPM study. Those projects were also solved to retrieve their respective loadings data. This saved time recreating two new projects from scratch. As in Ansys you can only go so far before there is not turning back.

The Static structural setup involved suppressing the other bodies except the con-rod, importing the loading data from the rigid dynamics, and setting the conrod to be a flexible part to model deformation. Also the mesh size for all the studies were set to 2mm. After this is the setup of the steady state thermal.

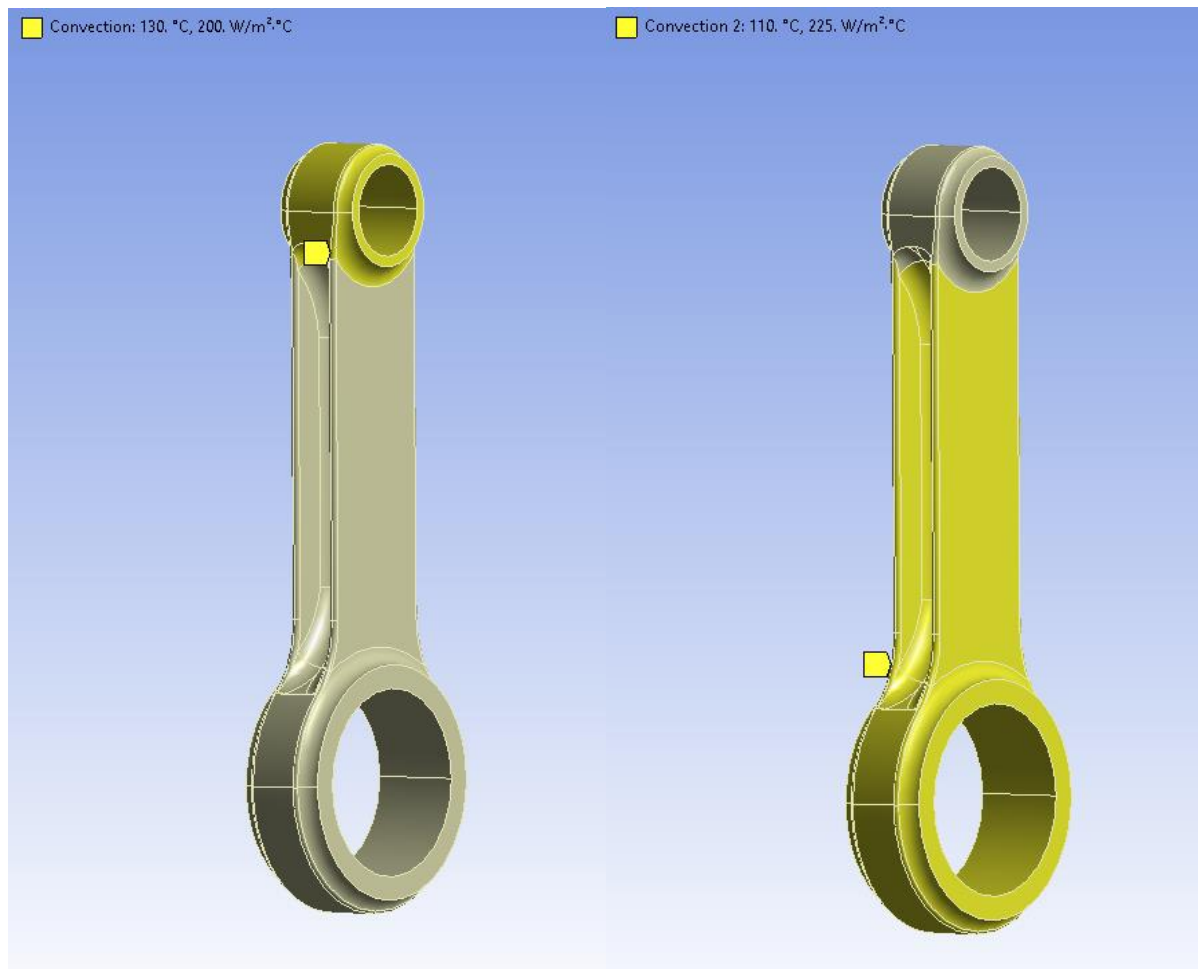


Figure 6 – Thermal loading

Again, the initial setup is quite the same as the static structural. For the thermal loading it is extremely difficult to get temperatures for a con rod under operating conditions. So, some educated guessed must be made. In this case two separate thermal loads were applied. A load of 130 degrees Celsius at the little end and 110 degrees Celsius at the big end up to the little end. The difference in temperature is due to the little end being closer to the area of combustion and the big end swimming in oil. The coefficients are based on that of a piston study being around 500W/m^2 . With steel having a lower thermal conductivity it is appropriate to lower the coefficient.

With this the thermal study can be solved for temperature, and the solution linked with the setup of the static structural. Thus, making a complete structural and thermal study.

The final setup in the static structural requires setting up the weak springs setting in the analysis settings. As well as applying an additional load to replicate the combustion pressure. This engine has been modelled in an engine simulator in which predicted combustion pressure has been estimated. This is the force that has been applied (**figure 7 see force F**). The forces seen in figure 7 are the ones generated at 8000RPM.

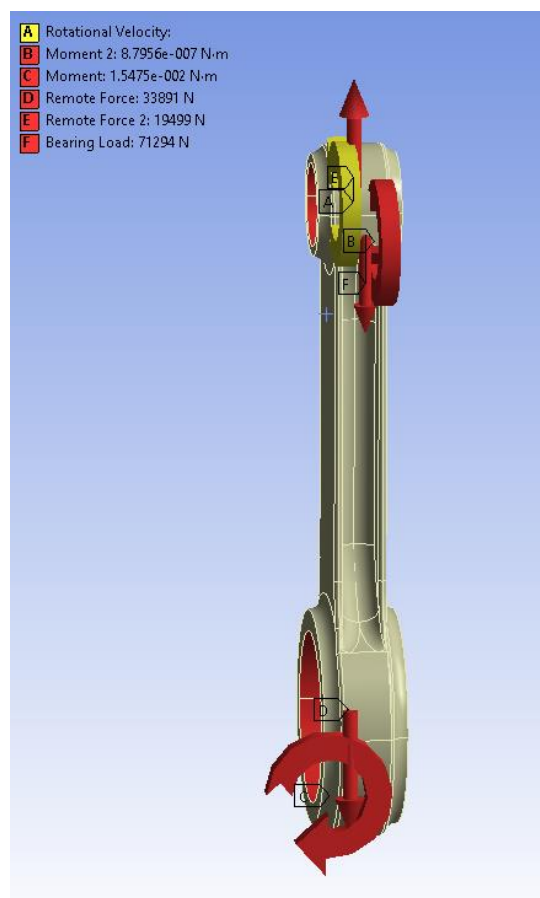


Figure 7 – Connecting rod loadings

Also, at each RPM range there will be two cases solved. one with the combustion force and another without. This is to get data that also replicates the top of the exhaust stroke. This is important as that is where the maximum tensile forces are seen on the conrod.

3 Test Results

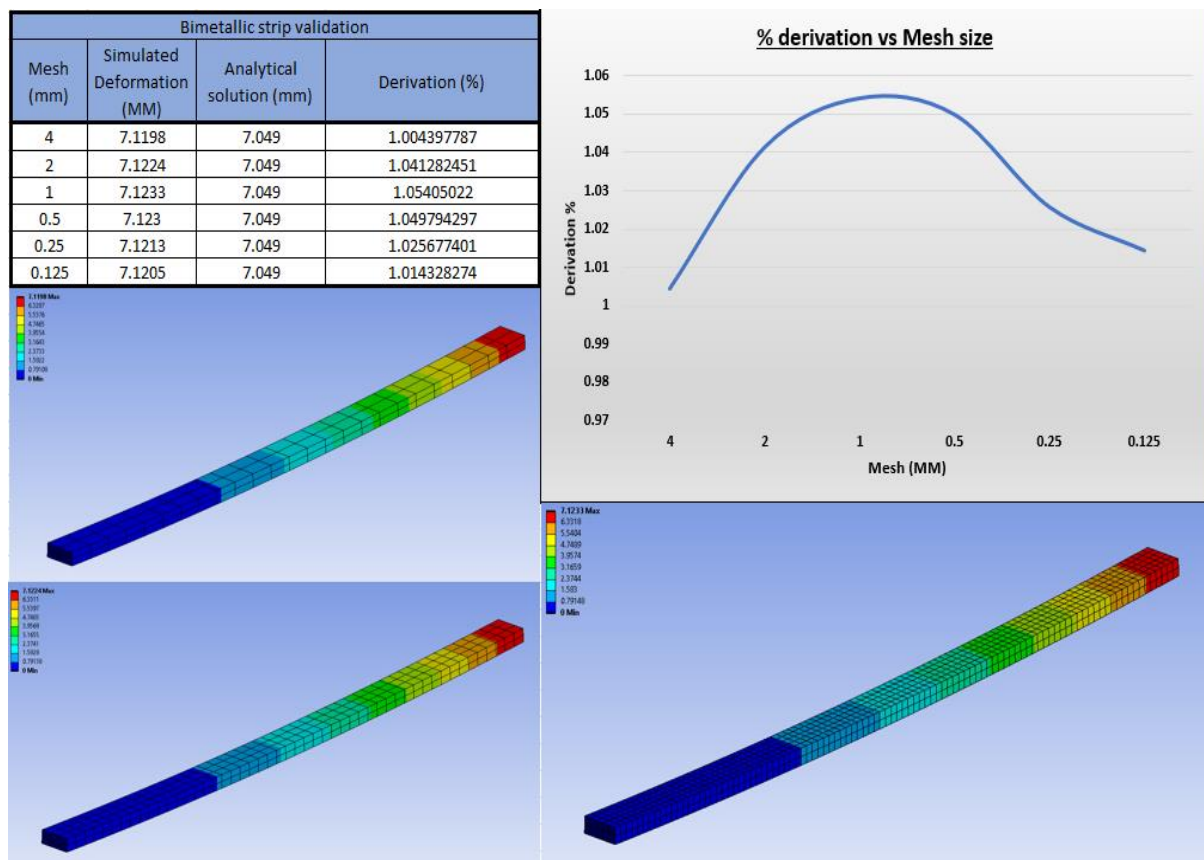


Figure 8 – Validation results



Figure 9 – Results from Ansys connecting rod study

4 Results Analysis

4.1 Validation Analysis

The validation proved that Ansys is Accurate to roughly 1% (**See Figure 8**). This inaccuracy is also an overestimation which is possibly a good thing. This makes it more likely that a component made in Ansys could be over engineered by 1%. But in cases where absolute accuracy is required then maybe Ansys is not the ideal solution to the problem. A number of mesh sizes were also tested, resulting in little change of improvement to the 1% inaccuracy. Although it seems that small improvements were made at the two extremes of the meshing densities.

4.2 Conrod study Analysis

Starting with the Deformation results. These results show very small amounts of deformation. The maximum being 0.19mm on the exhaust stroke at 10,000RPM. This result is interesting at it proves that the con-rod is susceptible to breaking under tensile conditions. Similar amounts if deformation were calculated at the 8000RPM range leading to question how well Ansys models more microscopic quantities. Either this or the deformation has been determined over a longer period of time than would be seen in the engine. the results from the elastic strain may back up this theory. As the Maximum strain at 8000RPM was calculated at 3.3mm of extension. if this happened in reality then the con-rod would stretch far enough at TDC to hit the head and or the valves. Again, this is likely attributed to Ansys applying the load over a prolonged period of time, unlike in the engine where the load is applied and removed in a fraction of a second.

The stress results are the most useful. These results allow for us to see the fatigue life of the conrod at each RPM Range. The fatigue life was calculated with the help of a study. In this study the fatigue life of 4340 steel was measured in a laboratory using a cyclic loading machine. A number of samples were tested to failure at different cyclic loadings to establish a failure life curve for 4340 steels (Çalışkan, Gürbüz, 2021).

With their failure life curve for 4340 steels, it was possible to approximate how many cyclic loadings the conrod could see before failure. Dividing that by RPM results in the amount of time before fatigue failure in minutes and seconds as a fraction (**See Figure 9**).

For the combustion loadings we determined that the con-rod would last indefinitely due to the loading not being high enough to reach fatigue point. However, fatigue can be seen on the exhaust stroke. And since the engine exhausts every other revolution then that doubles the fatigue life of the conrod within the engine. Since at 8000RPM the engine will exhaust 4000 times per minute.

The final interesting result that can be obtained was at what RPM limit will the conrod see no fatigue. According to (Çalışkan, Gürbüz, 2021) fatigue starts at roughly 600MPa for 4340 steels. So, if a scatter graph is plotted showing the maximum stresses against RPM, then a polynomial trend line can be plotted so the non fatigue RPM limit can be found (**See Figure 9**). Looking at the Graph in figure 9 it can be seen that fatigue begins at roughly 7700RPM.

This Data is extremely useful. This allows for the engines RPM limit to be set with the life of the components in mind. As well as determining if it is worth pushing the component to a point where fatigue becomes more likely. For example, if it happened that changing gear was ideal at 8000 RPM and would make the difference in a race, then you would know that the engine can only be raced for so much time before a rebuild. This would help a team decide how often that component requires replacement and if that can be afforded.

Ansys Workbench also allows for design improvements to be made. Such as the stresses seen on the part in **figure 15, 21 and 27**. These indicate that the component would likely fail on the lower part for the rod near the big end. To alleviate this stress improvements could be made by reducing the mass of the piston, if possible. And or strengthening the con-rod on that lower section. Although This would only be desired if there is potential for more power to be made at a higher RPM range.

5 Conclusion

from this study, it is clear that Ansys Workbench is a great tool for engineering design. In the Validation it is true that it can be used to solve complex static problems to a good level of accuracy. It is also proven to be an informative tool for the structural limitations of the Performance Carrillo connecting rod. Workbench can also be a great tool for designing a part of a complex geometry which would otherwise be impossible calculate for its structural suitability. But there are draw backs.

These drawbacks include its steep learning curve, long setup process and its inability to solve a multi body problem. The steep leaning curve comes from its clunky interface. This makes learning the location of settings an overwhelming and slow process. The long setup time can also be attributed to the clunky styles of the interface. If this is continued to be streamlined, then it could be a better software package. Its inability to solve a multi body problem, such as an assembly currently is not the fault of the software. To solve the stresses of an entire assembly in a reasonable time would require a room full of super computers. without this ability you must check each component individually. Making for a slow design process for a single person.

This inability to simulate the stresses on an assembly or unpredictable conditions often lead to over engineering. And where a component must be light, they are often subject to destructive testing. Or torture testing. Destructive testing is common practice especially in the aeronautical industry.

Despite the shortcomings of Ansys workbench, it is an effective software for what it is made to do. As proven in this report and by others who have achieved comparable results to real test data. Such as one thermal study where they found just a 5% difference in the results (A Sahoo Et al, 2021).

6 Bibliography

- Dubal Software, (2018), 0078 – Bimetallic strip, Accessed Via: Microsoft Teams, Accessed: 08/12/2022.
- D, Creason, (2013), Connecting Rod Tech: Forged and Billet Steel Rods, enginelabs, Enginelabs.com, Available at: <https://www.enginelabs.com/tech-stories/connecting-rod-tech-forged-and-billet-steel-rods/#:~:text=The%20most%20common%20type%20of,heat%20treatment%20applied%20to%20it.>, Accessed: 17/12/22.
- Matweb, AISI 4340 Steel normalized, 25 mm round, Available at: <https://www.matweb.com/search/DataSheet.aspx?MatGUID=fd2df45bffa54018b54989bc14092d9f&ckck=1>, Accessed: 08/18/2022.
- Matweb, Aluminium 2618-T61, Available at: https://www.matweb.com/search/datasheet_print.aspx?matguid=f6d0bebbfc7248838243b7fa141431ba, accessed: 18/12/2022.
- CP-Carrillo, Frequently Asked Questions – Pistons, Available at: <https://www.cp-carrillo.com/ft-2406-frequently-asked-questions-pistons.html#:~:text=Both%20CP%20CARRILLO%20and%20BULLET,3%20days%20to%20your%20specification.>, Accessed: 09/12/2022.
- M, Veldhuijzen, (2015), Supermono documentary Mike Velthuijzen, European Supermono Cup, YouTube.com, Available at: <https://www.youtube.com/watch?v=coBIDeEXUhM&t=299s>, Accessed: 18/12/22.
- Motorcyclespecs, (1997), Yamaha SZR 660, Motorcyclespecs.com, Available at: https://www.motorcyclespecs.co.za/model/yamaha/yamaha_szr660%2095.htm, Accessed: 18/12/22.

- S, Çalışkan, R, Gürbüz, (2021), Determining the endurance limit of AISI 4340 steels in terms of different statistical approaches, Department of Metallurgical and Materials Engineering, Middle East Technical University, Available at: <https://www.uwtsd.ac.uk/library/>, Accessed:10/12/2022.
- A Sahoo, R Nayak, A Senapati, M, Roul, (2021), Validation of experimental results with theoretical by using ANSYS workbench on vertical tube subjected to natural convection heat transfer without internal obstacles, Department of Mechanical Engineering, GIET University, Gunupur, Available at: <https://www-sciencedirect-com.ezproxy.uwtsd.ac.uk/science/article/pii/S2214785321070851>, Accessed:18/12/22.

7 Appendices

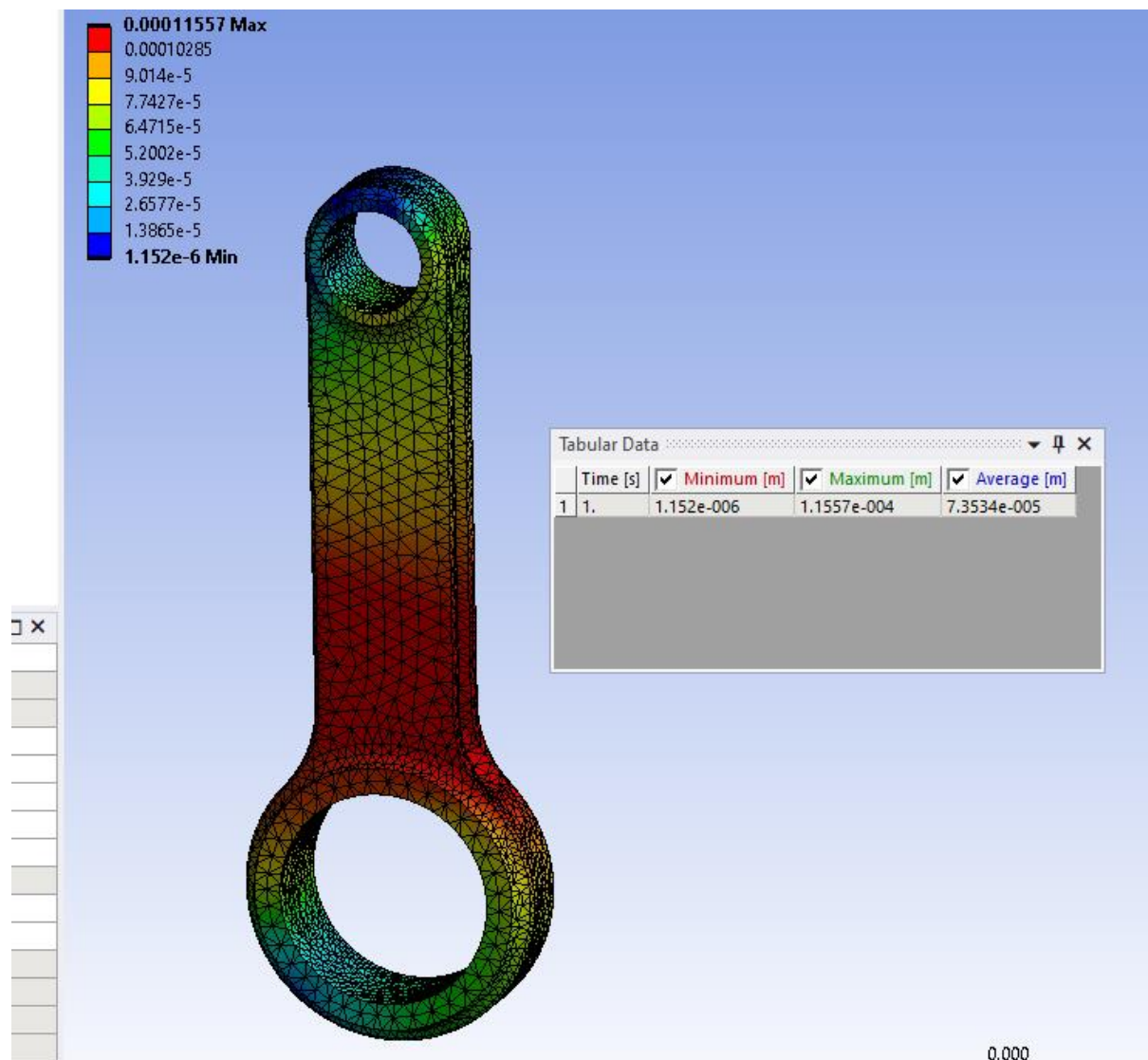


Figure 10 – Total Deformation 8000RPM (with combustion pressure)

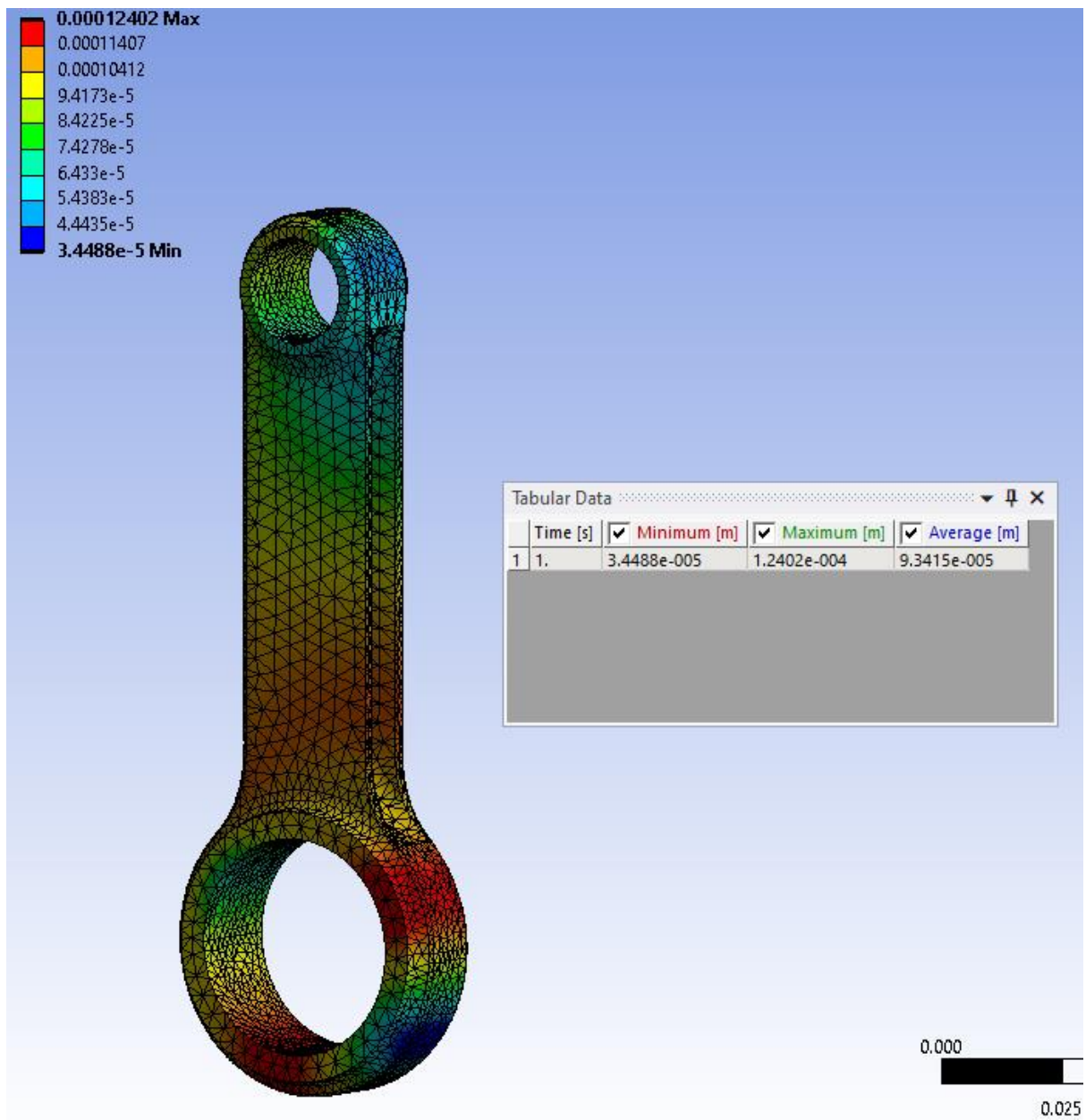


Figure 11 – Total Deformation 8000RPM (Exhaust Stroke)

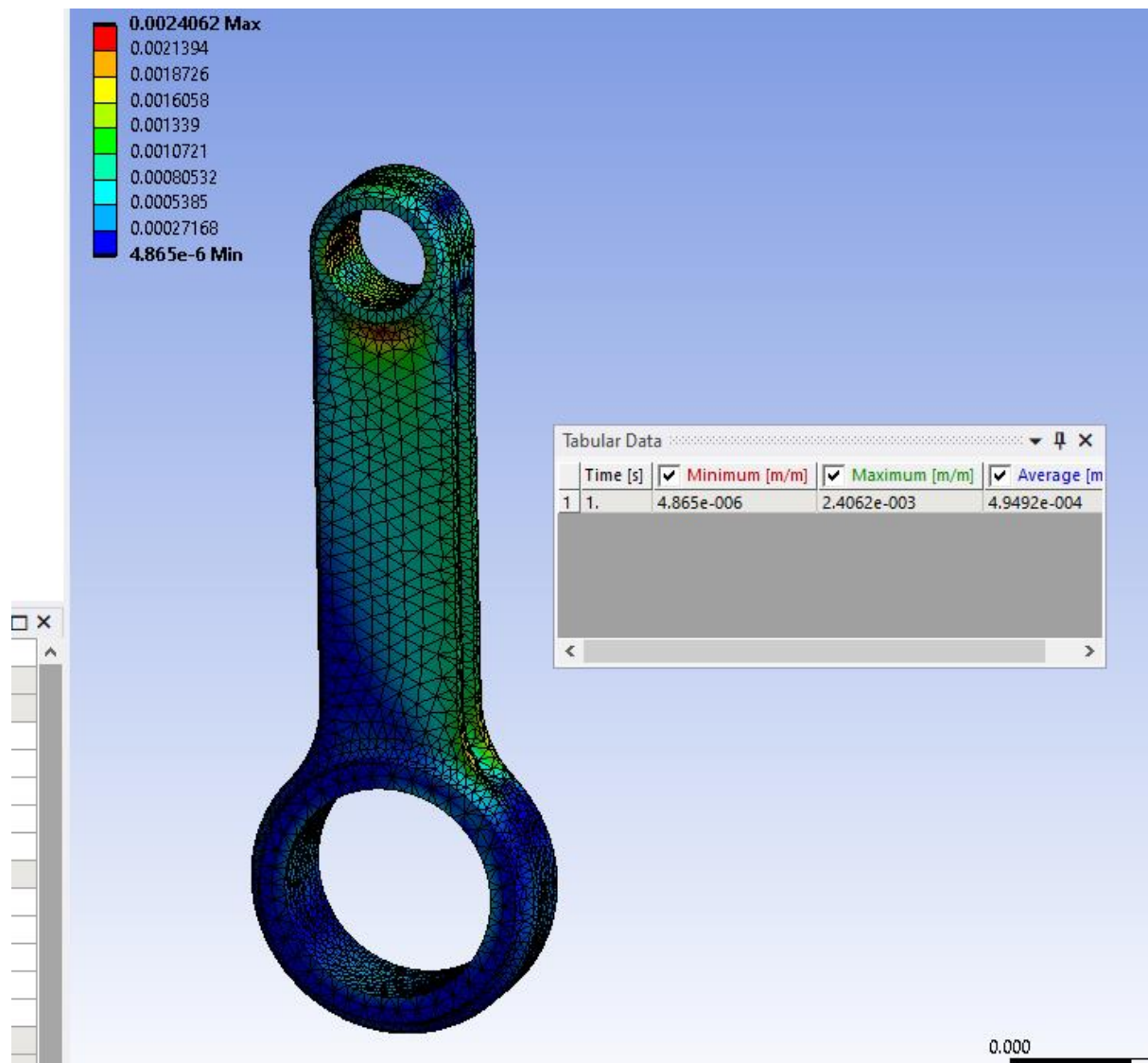


Figure 12 – Elastic strain 8000RPM (With combustion force)

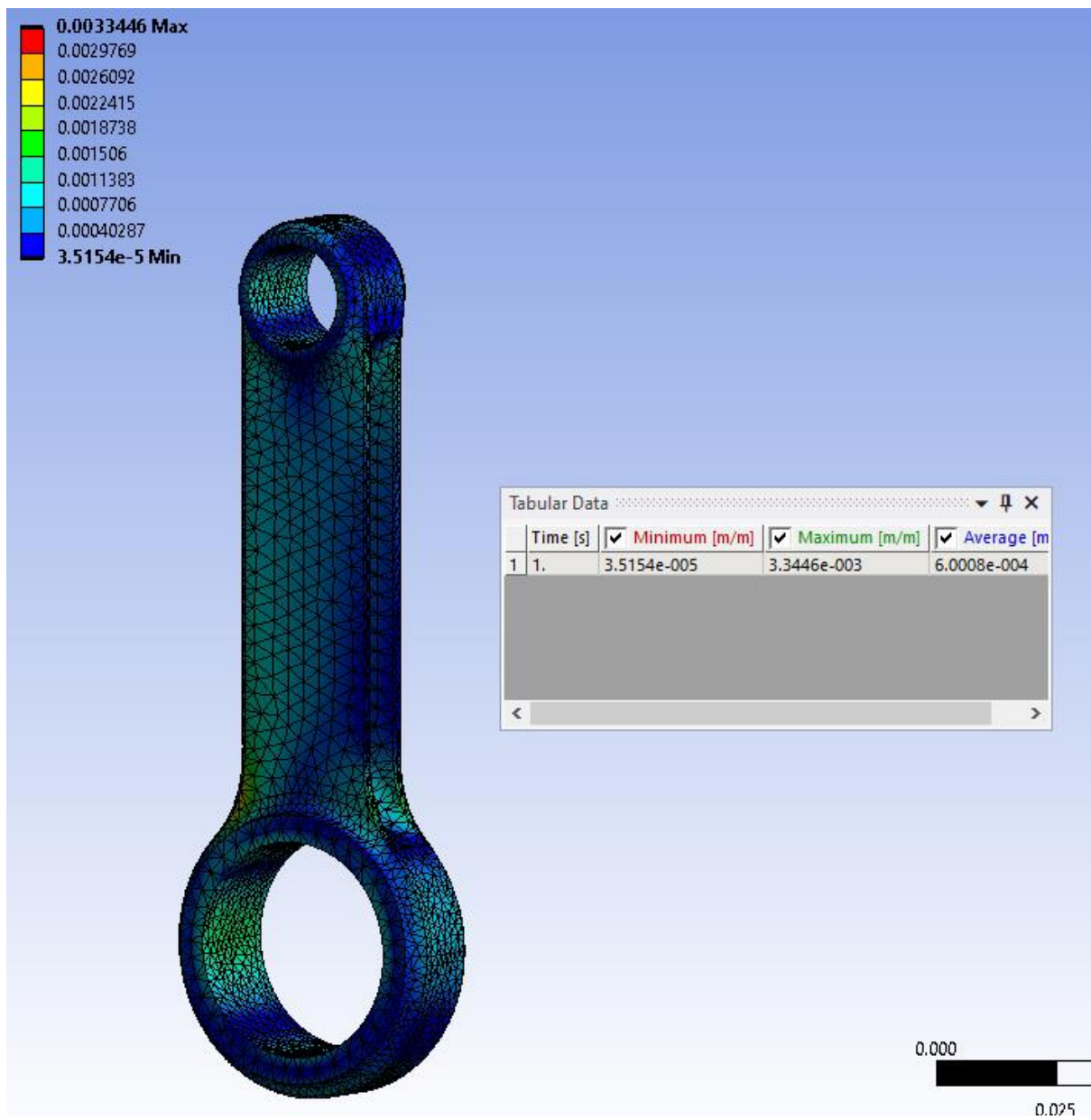


Figure 13 – Elastic strain 8000RPM (Exhaust stroke)

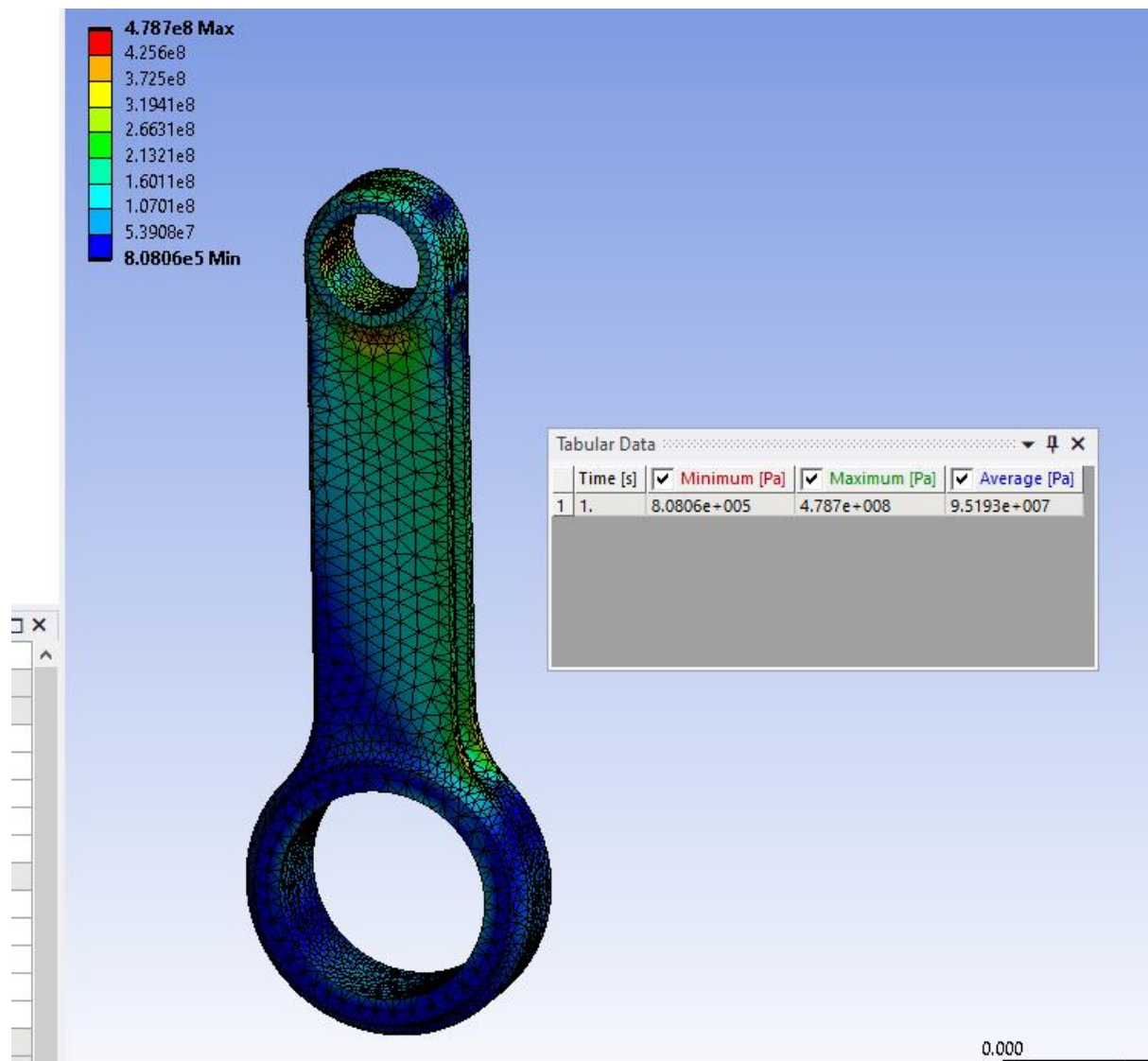


Figure 14 – Stress 8000RPM (With combustion force)

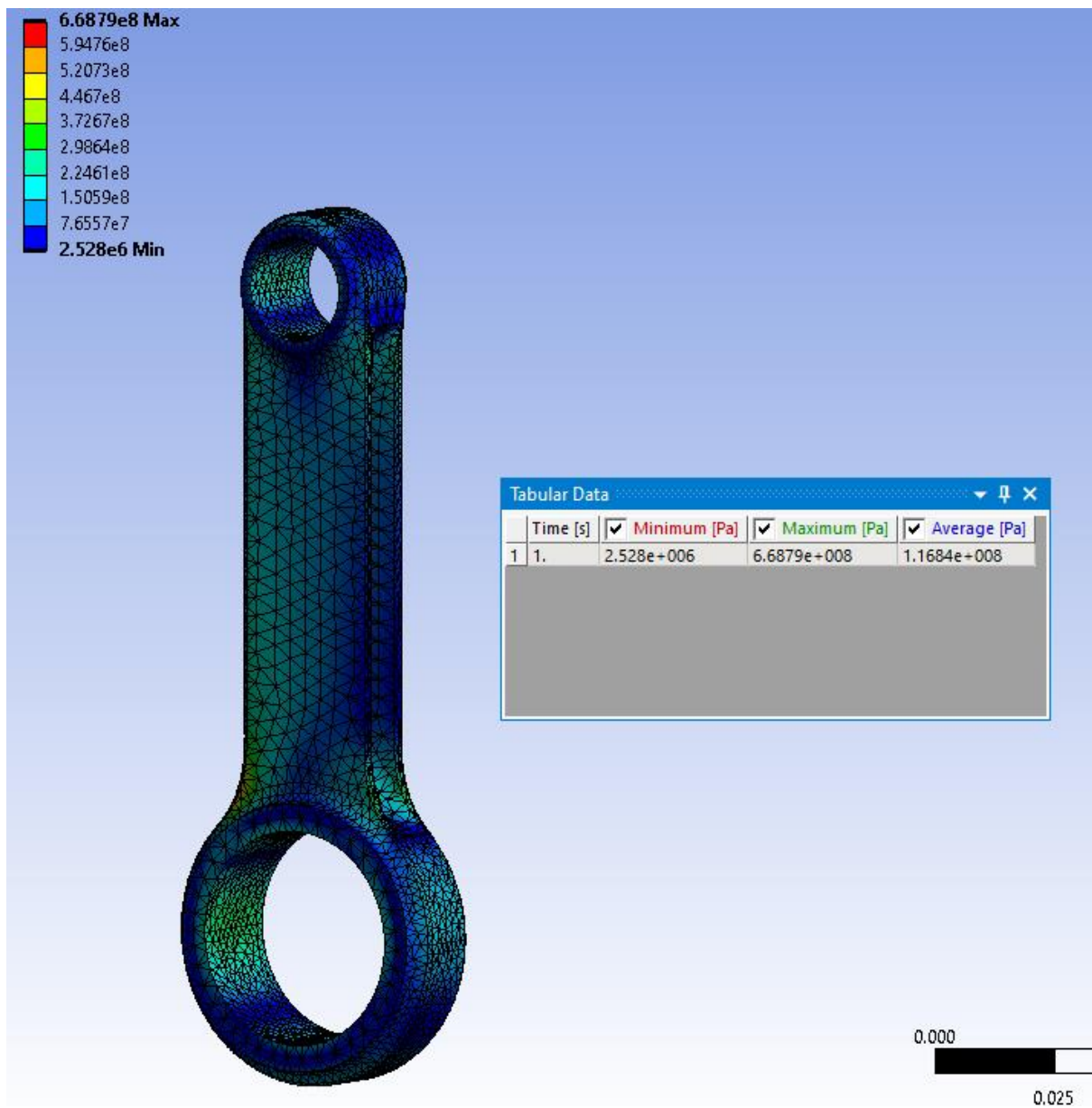


Figure 15 – Stress 8000RPM (Exhaust stroke)

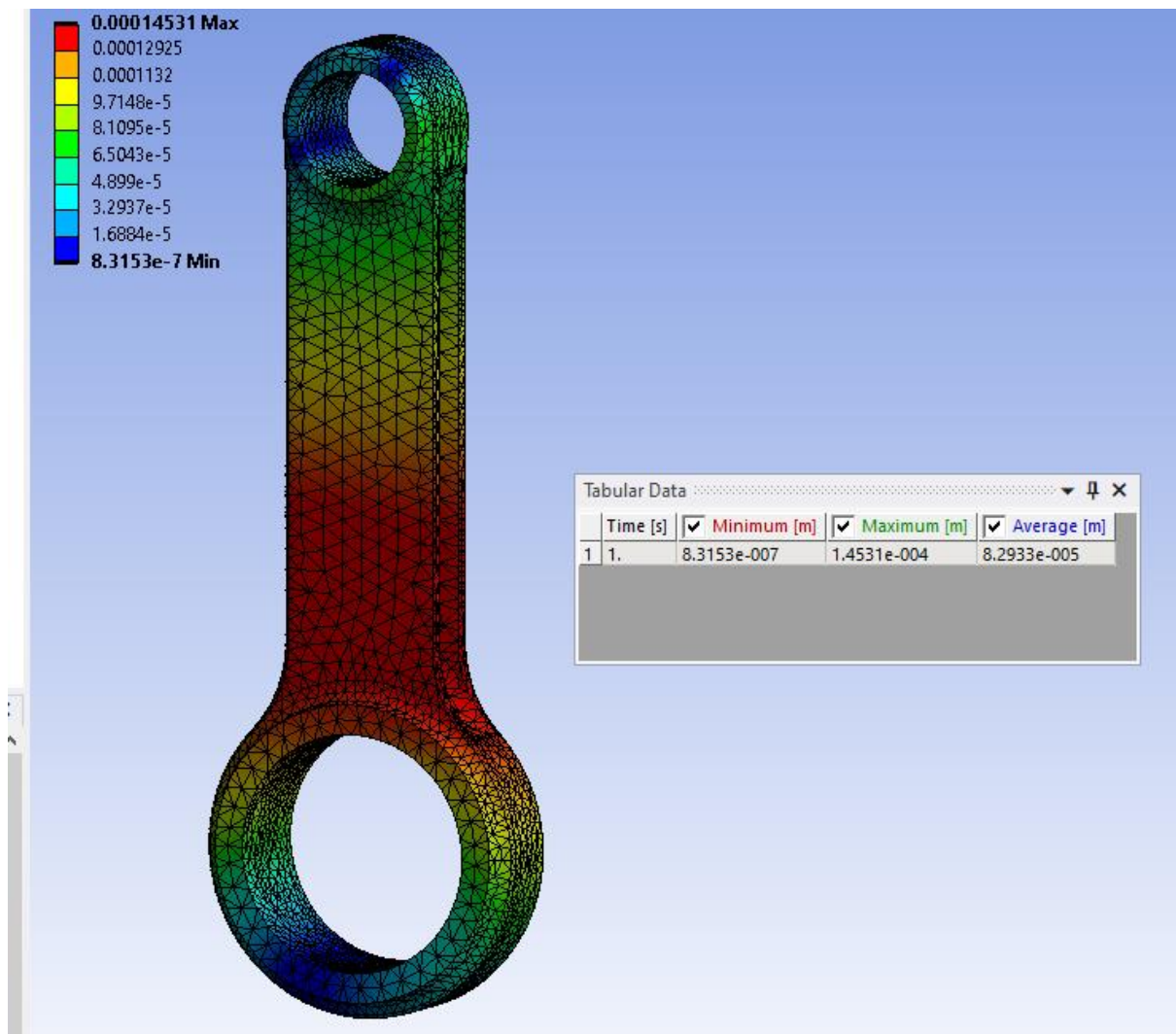


Figure 16 – Total Deformation 9000RPM (with combustion pressure)

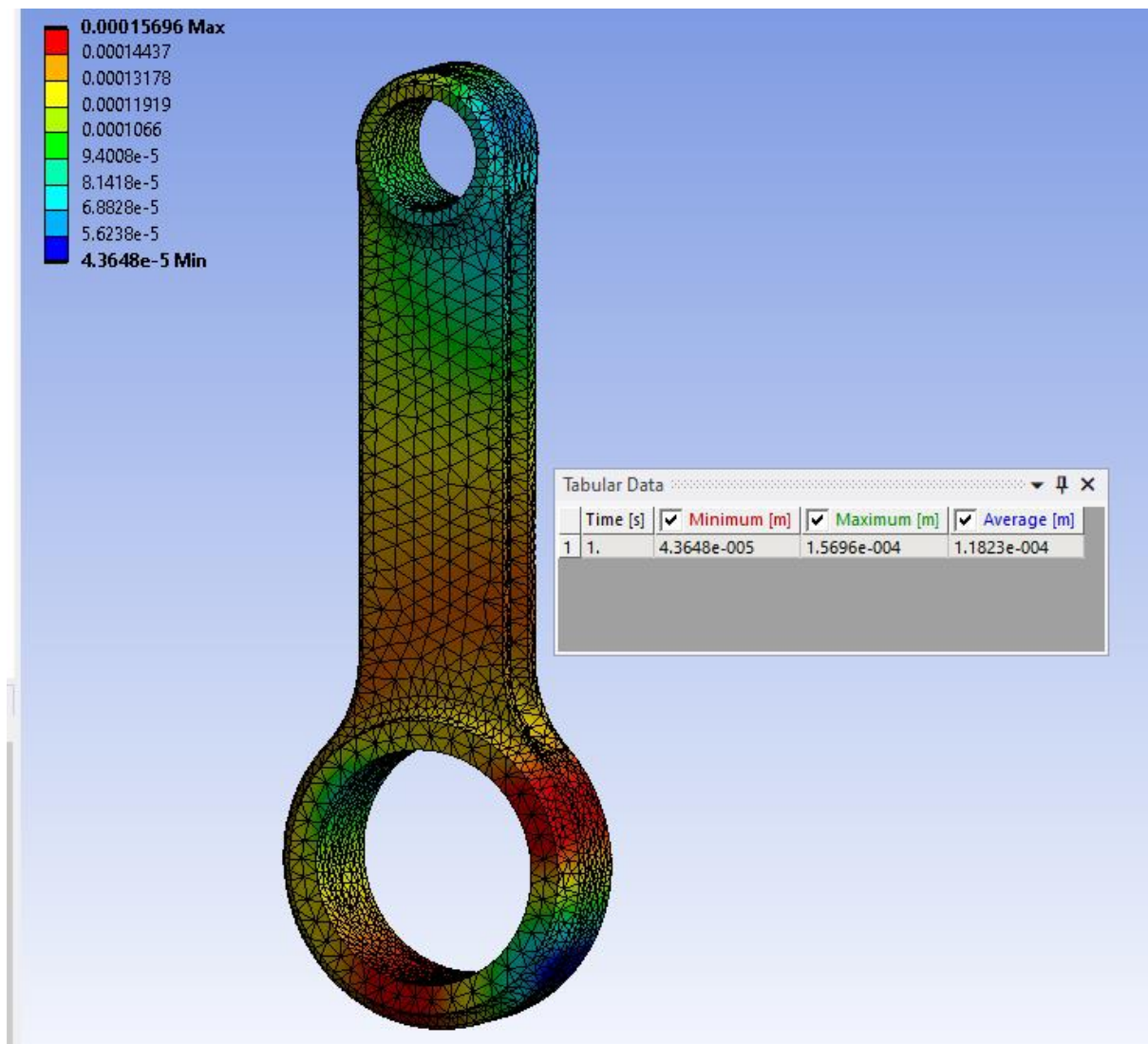


Figure 17 – Total Deformation 9000RPM (Exhaust stroke)

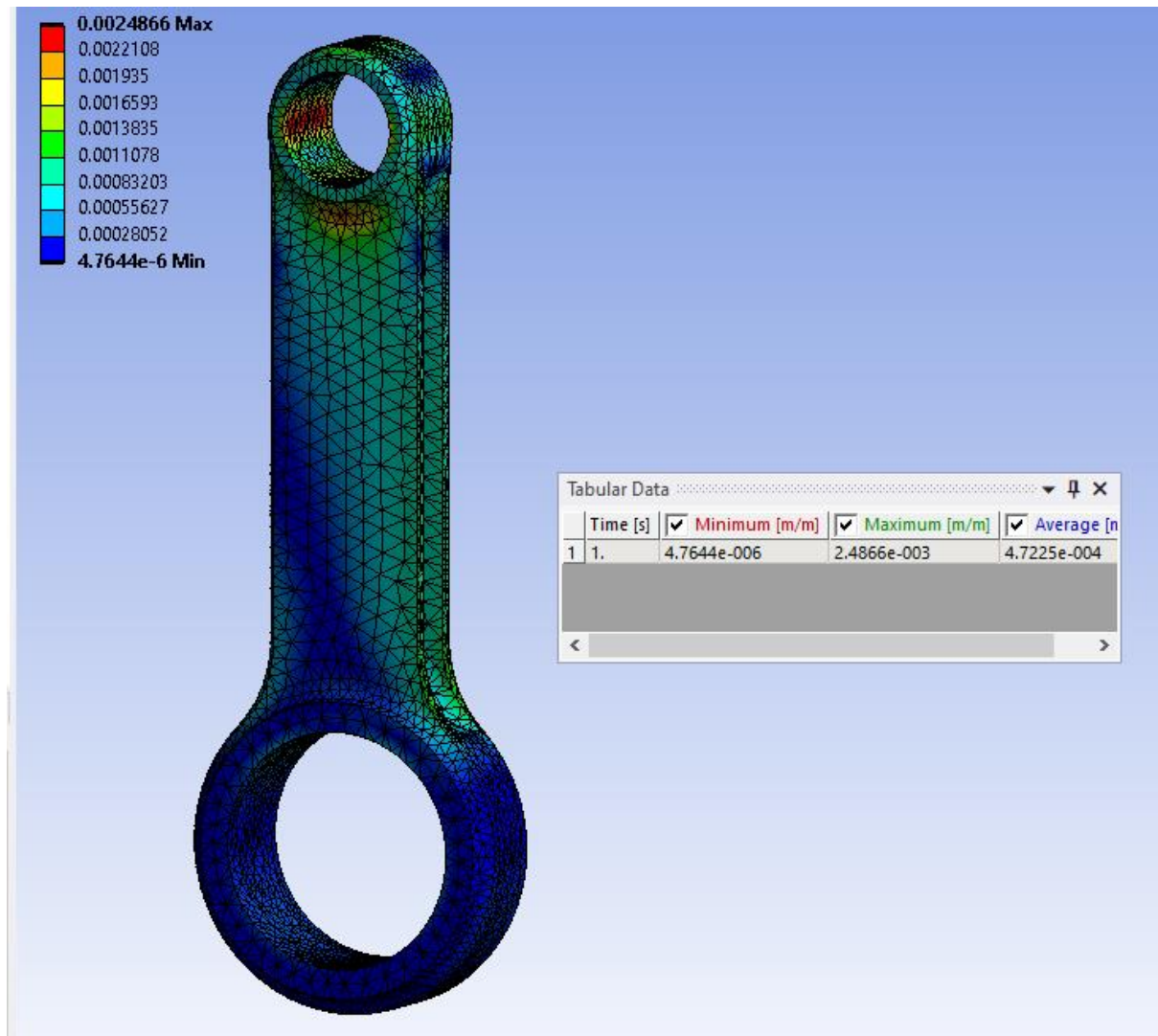


Figure 18 – Elastic strain 9000RPM (With combustion Pressure)

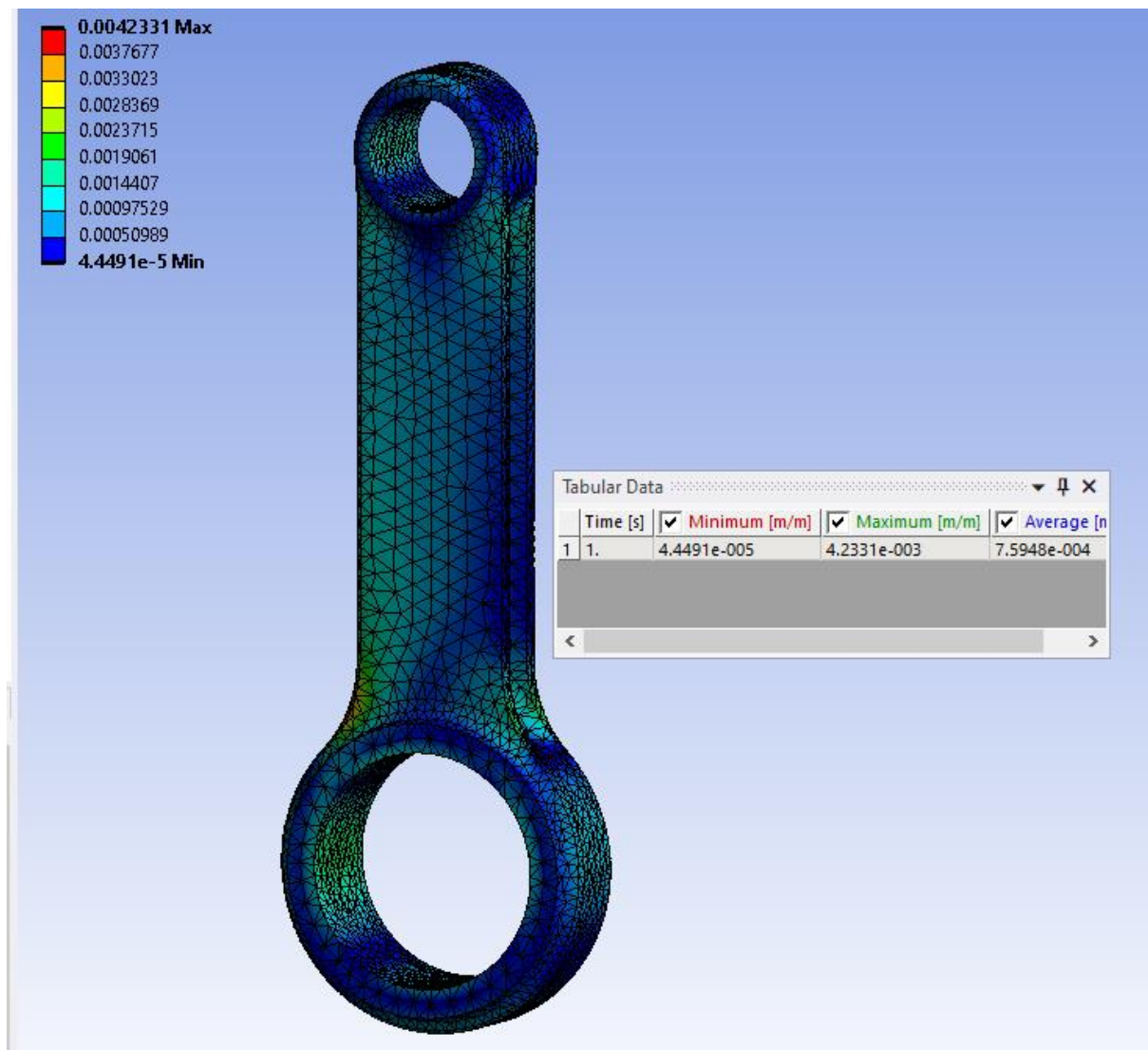


Figure 19 – Elastic strain 9000RPM (Exhaust stroke)

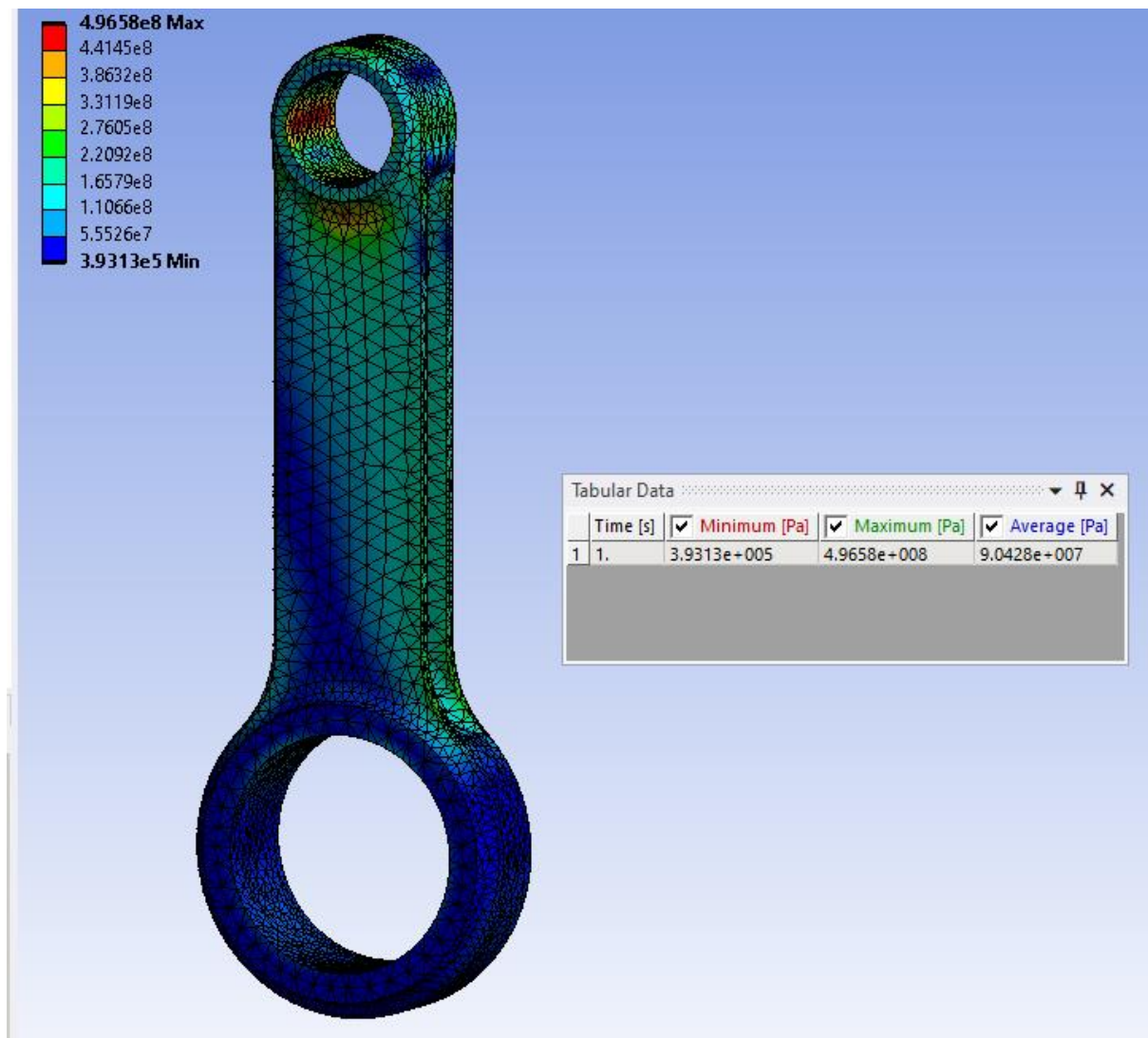


Figure 20 – Stress 9000RPM (With Combustion pressure)

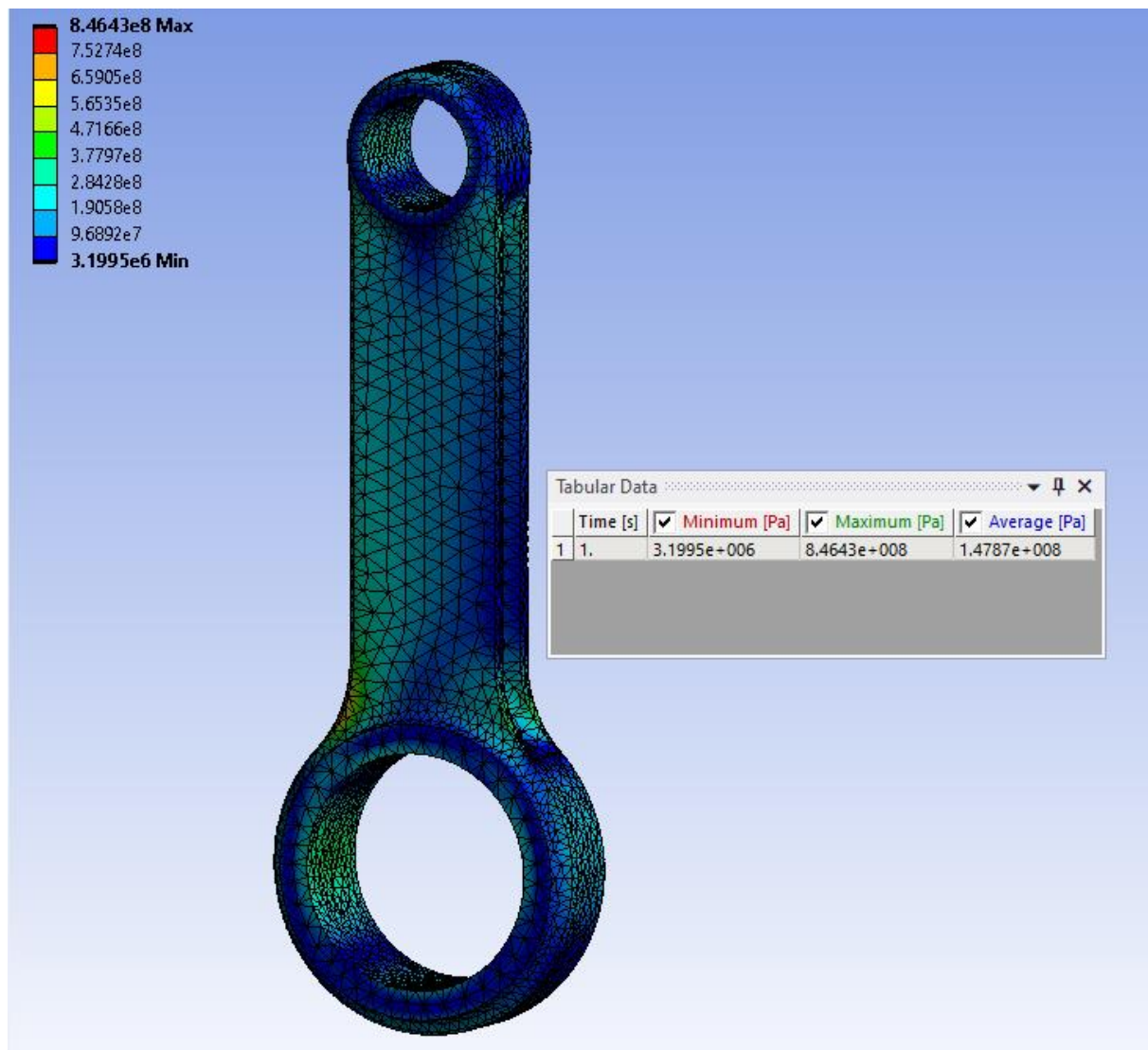


Figure 21 – Stress 9000RPM (Exhaust stroke)

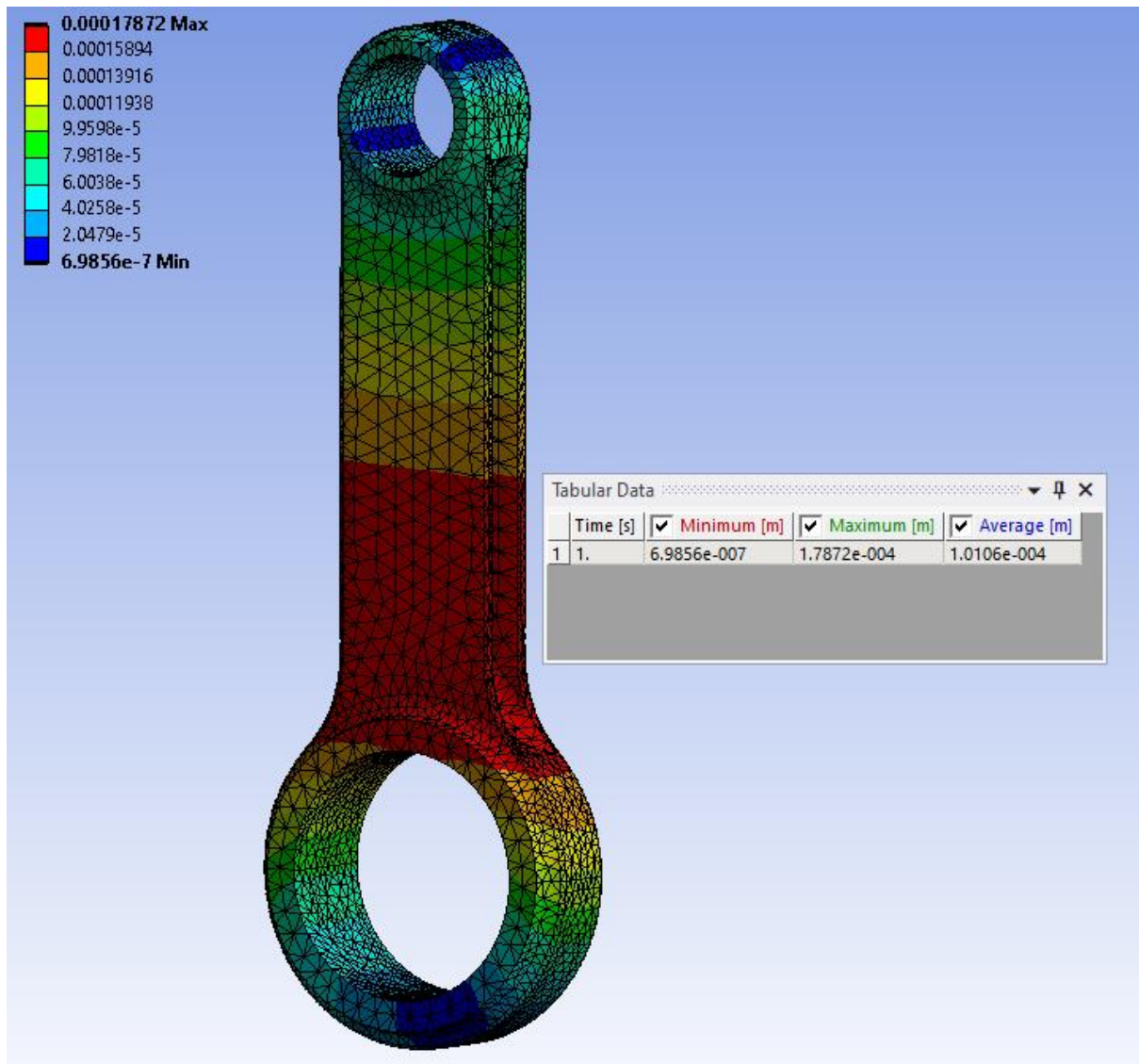


Figure 22 – Total Deformation 10,000RPM (With Combustion Pressure)

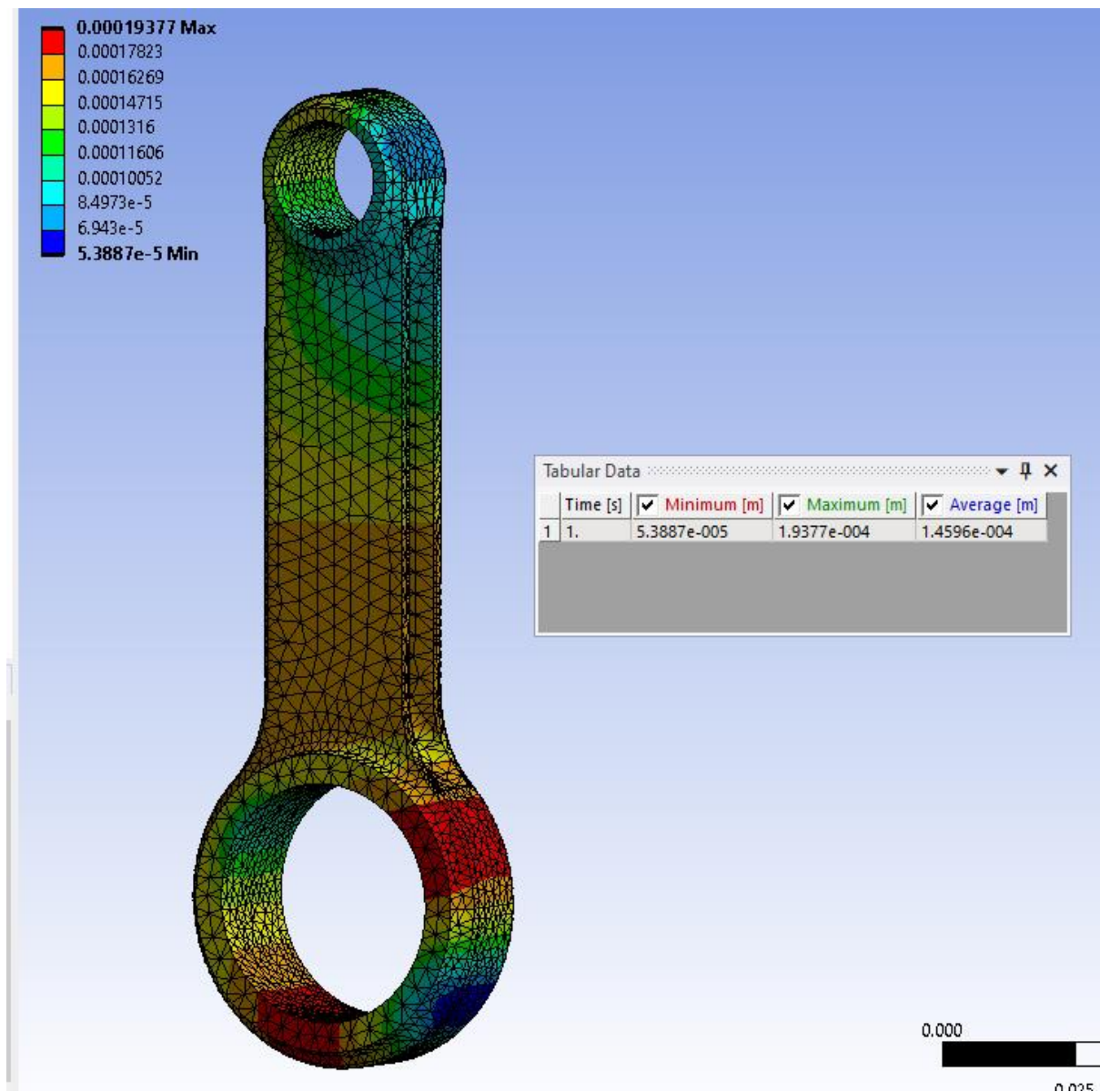


Figure 23 – Total Deformation 10,000RPM (Exhaust stroke)

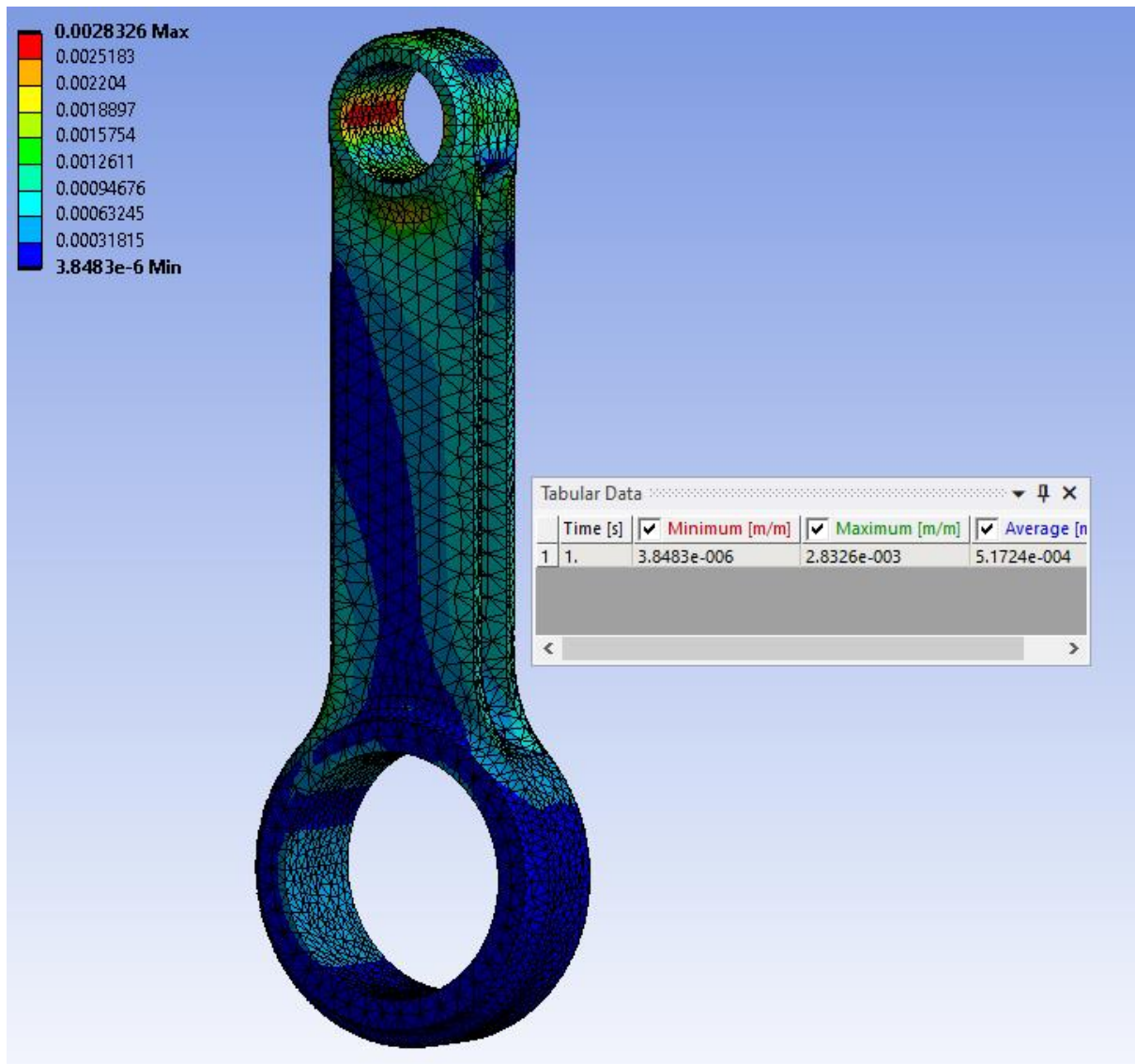


Figure 24 – Elastic strain 10,000RPM (With combustion pressure)

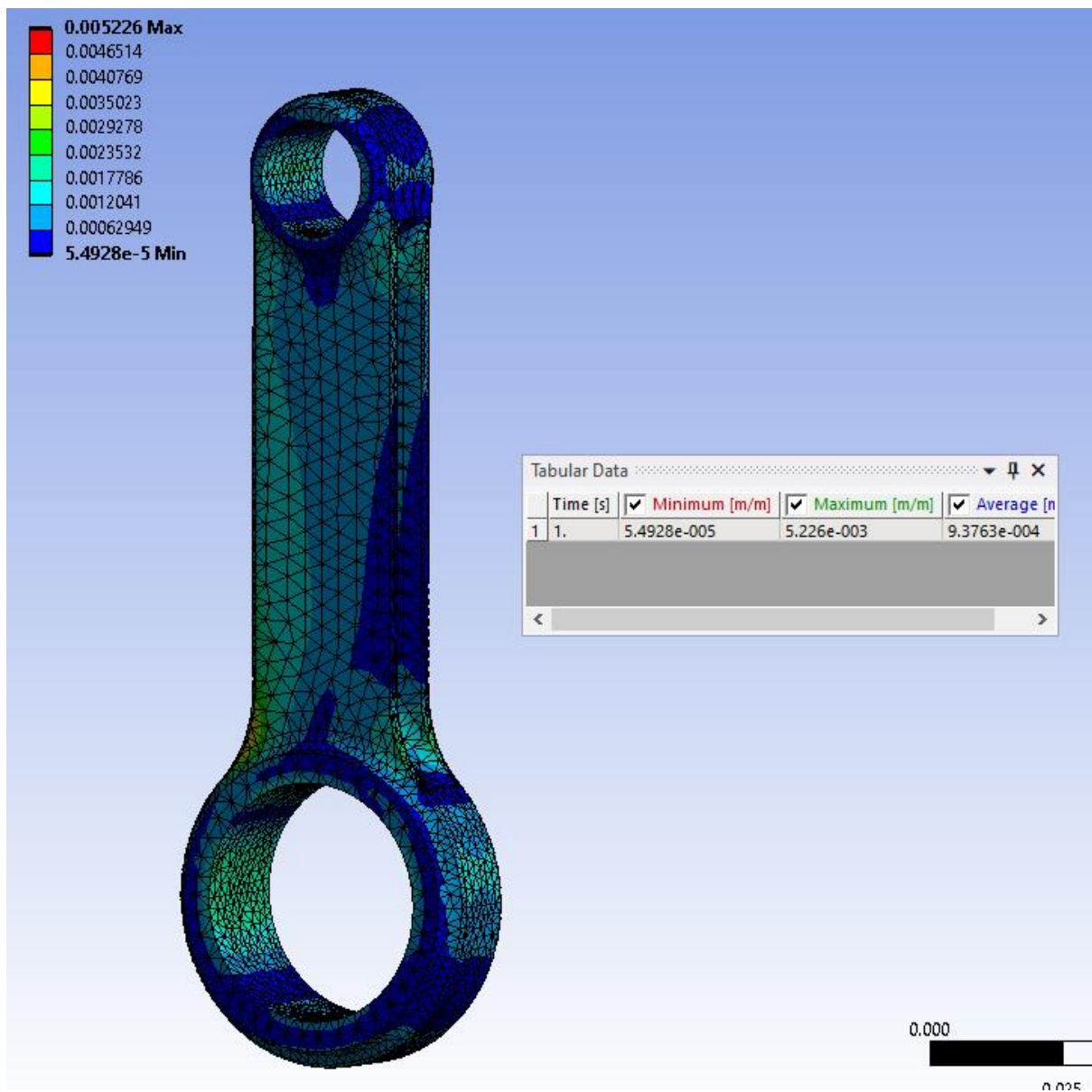


Figure 25 – Elastic strain 10,000RPM (Exhaust stroke)

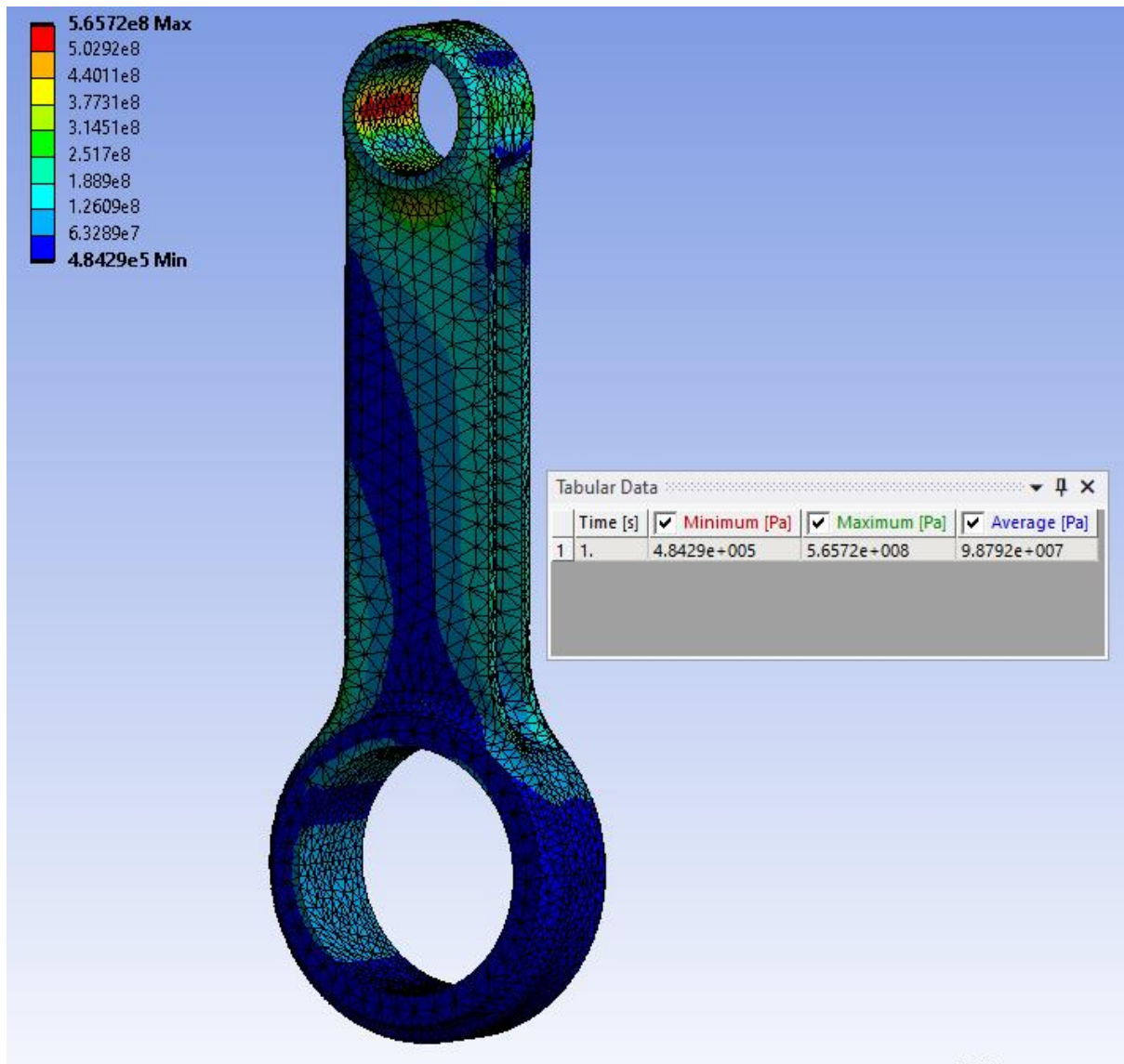


Figure 26 – Stress 10,000RPM (with combustion pressure)

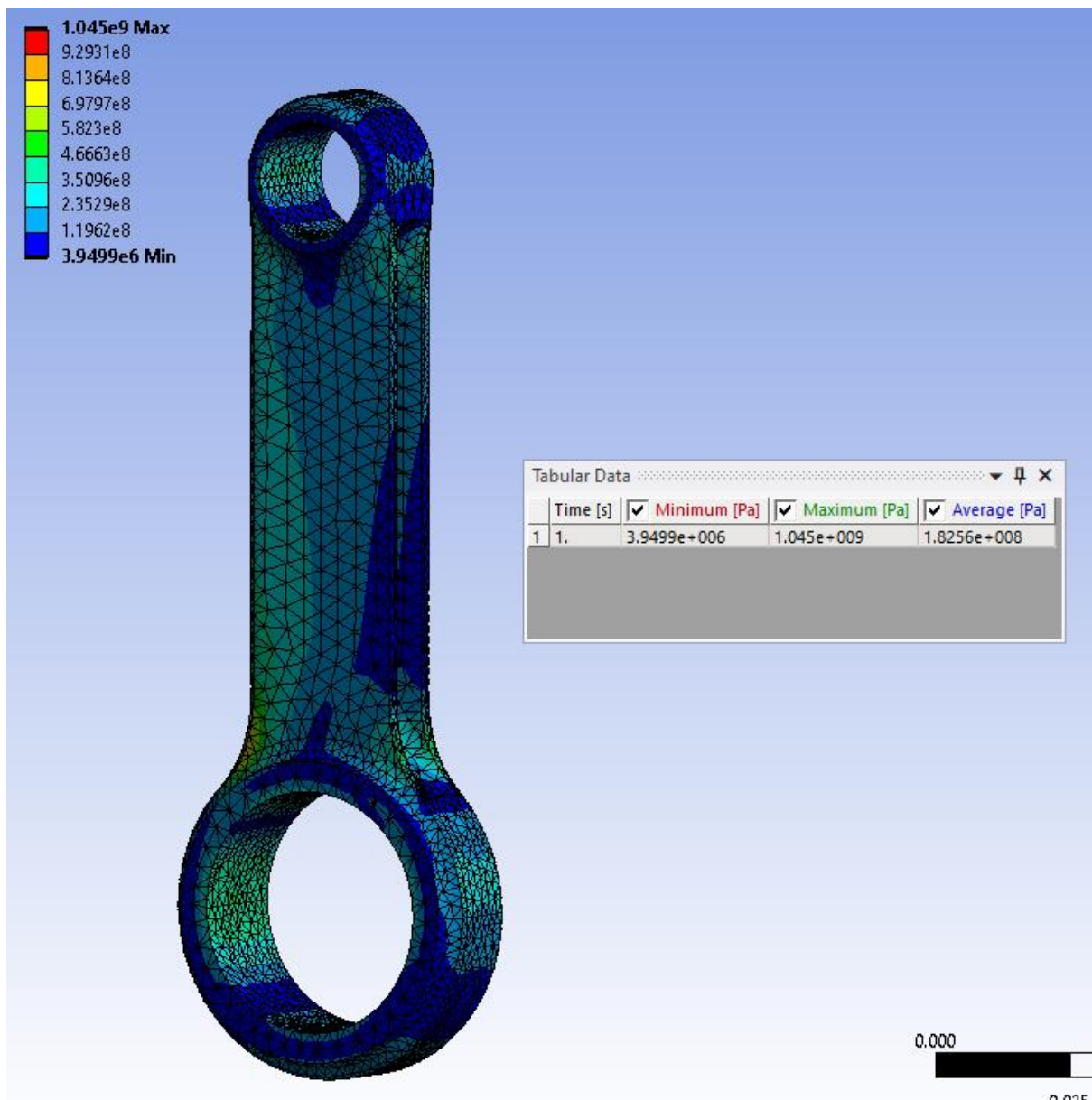


Figure 27 – Stress 10,000RPM (Exhaust stroke)