逢甲大學學生報告 ePaper

Stress Analysis of Piston

時凡浩 D9221660,盧亭君 D9270457 are Junior students in Dept. Aerospace & System Engineering

Hsien-Chie Cheng(鄭仙志)

Computer aided engineering analysis-Nastran/Pastran

94 school year 1st semester

Index

中文摘要與關鍵字

由於電腦科技及軟體工程之發達，電腦輔助工程技術已經成為國內甚至全世界業界產品設計、分析及製造之基礎。同組二人於**本學期修習逢甲大學航太與系統工程學系鄭仙志老師開設之**電腦輔助工程-***NASTRAN***，學習有效地運用這些先進電腦輔助工程技術以輔助其產品設計，目的可大量縮短產品開發乃至最終上市之時間，進而提升產品之競爭性。修習電腦輔助工程-***NASTRAN*** 的課程，透過基本電腦輔助工程分析技術及有限元素法的學習，運用 ***MSC/NASTRAN*** 商用有限元分析軟體及 ***MSC/PATRAN*** 前後處理器之操作方法，含幾何建置、前處理、結構分析及後處理等四步驟，並進行各式工程結構靜、動態彈性行為的分析。

配合二年級下學期修習鄭老師開設之電腦三維模型建構-PRO/E 課程，運用二維的草繪圖為出發點，進而利用三維功能使其製作成零組件，最後將各零組件組裝為物件，設計一套汽車引擎內之活塞組，再運用有限元分析軟體 MSC/NASTRAN 及 MSC/PATRAN 分析同一物件但設定不同材料後其活塞發動時其材料受力、受力後活塞位移（形變）、壓力變化…。依據分析結果，再修改原模型使活塞達到最有效之改善。

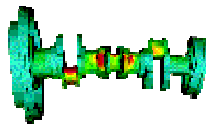
關鍵字

電腦輔助工程-NASTRAN、MSC/NASTRAN、MSC/PATRAN、有限元素法、電腦三維模型建構-PRO/E、活塞、材料受力、受力後活塞位移（形變）、壓力變化

Conspectus

What is FEA?

Finite element analysis was first developed for use in the aerospace and nuclear industries where the safety of structures is critical. Today, the growth in usage of the method is directly attributable to the rapid advances in computer technology in recent years. As a result, commercial finite element packages exist that are capable of solving the most sophisticated problems, not just in structural analysis, but for a wide range of phenomena such as steady state and dynamic temperature distributions, fluid flow and manufacturing processes such as injection molding and metal forming.



FEA consists of a computer model of a material or design that is loaded and analyzed for specific results. It is used in new product design, and existing product refinement. A company is able to verify that a proposed design will be able to perform to the client's specifications prior to manufacturing or construction. Modifying an existing product or structure is utilised to qualify the product or structure for a new service condition. In case of structural failure, FEA may be used to help determine the design modifications to meet the new condition.

Mathematically, the structure to be analyzed is subdivided into a mesh of finite sized elements of simple shape. Within each element, the variation of displacement is assumed to be determined by simple polynomial shape functions and nodal displacements. Equations for the strains and stresses are developed in terms of the unknown nodal displacements. From this, the equations of equilibrium are assembled in a matrix form which can be easily be programmed and solved on a computer. After applying the appropriate boundary conditions, the nodal displacements are found by solving the matrix stiffness equation. Once the nodal displacements are known, element stresses and strains can be calculated

within each of these modeling schemes, the programmer can insert numerous algorithms (functions) which may make the system behave linearly or non-linearly. Linear systems are far less complex and generally ignore many subtleties of model loading & behavior. Non-linear systems can account for more realistic behavior such as plastic deformation, changing loads etc. and is capable of testing a component all the way to failure.

Despite the proliferation and power of commercial software packages available, it is essential to have an understanding of the technique & physical processes involved in the analysis. Only then can an appropriate & accurate analysis model be selected, correctly defined and subsequently interpreted.

What's the difference between FEM & FEA ?

This is a very contentious issue, one that academics love to debate over a cool long-neck of a friday evening. I am going to stick my head on the block here & try to explain the difference, happy chopping my academic friends.

The terms 'finite element method' & 'finite element analysis' seem to be used interchanably in most documentation, so the question arises is there a difference between FEM & FEA ? The answer is yes, there is a difference, albeit a subtle one that is not really important enough to loose sleeps over.

The finite element method is a mathematical method for solving ordinary & elliptic partial differential equations via a piecewise polynomial interpolation scheme. Put simply, FEM evaluates a differential equation curve by using a number of polynomial curves to follow the shape of the underlying & more complex differential equation curve. Each polynomial in the solution can be represented by a number of points and so FEM evaluates the solution at the points only. A linear polynomial requires 2 points, while a quadratic requires 3. The points are known as node points or nodes. There are essentially three mathematical ways that FEM can evaluate the values at the nodes, there is the non-variation method (Ritz), the residual method (Galerkin) & the variation method (Rayleigh-Ritz).

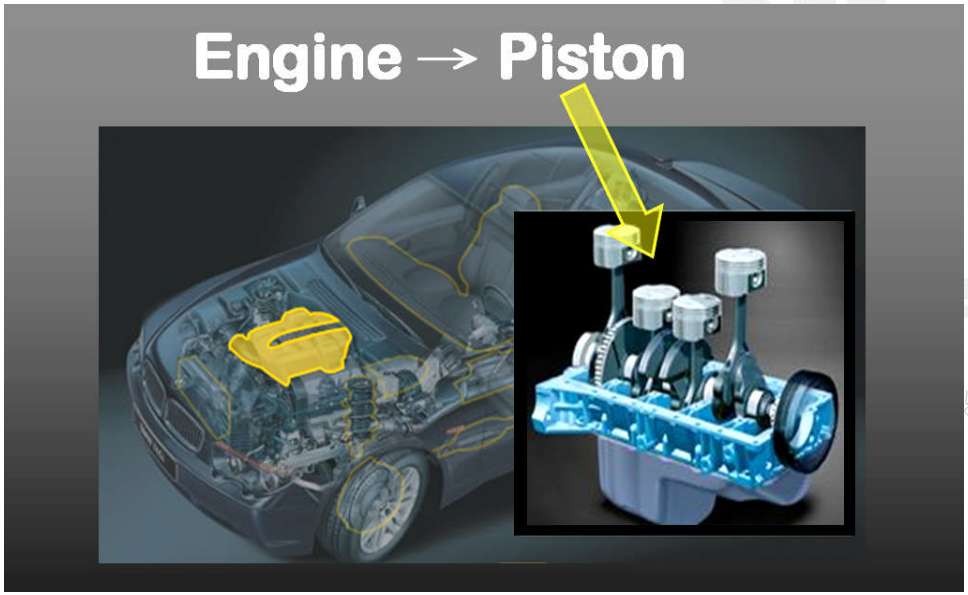
FEA is an implementation of FEM to solve a certain type of problem. For example if we were intending to solve a 2D stress problem. For the FEM mathematical solution, we would probably use the minimum potential energy principle, which is a variation solution. As part of this, we need to generate a suitable element for our analysis. We may choose a plane stress, plane strain or an ax symmetric type formulation, with linear or higher order polynomials. Using a piecewise polynomial solution to solve the underlying differential equation is FEM, while applying the specifics of element formulation is FEA, e.g. a plane strain triangular quadratic element.

Prolegomena

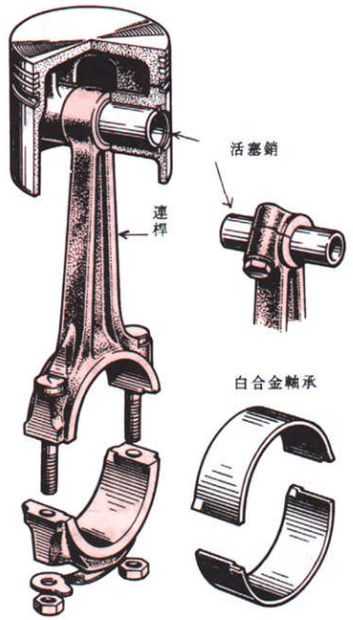
Base on the skills we learned how to build a model at the class "Solid modeling using Pro/E", in this semester we try to analysis the piston of car that we have seen in our daily life. And try to put it in different materials and justice out why the sport car has been so absorbed in there engine study.

Technological processes

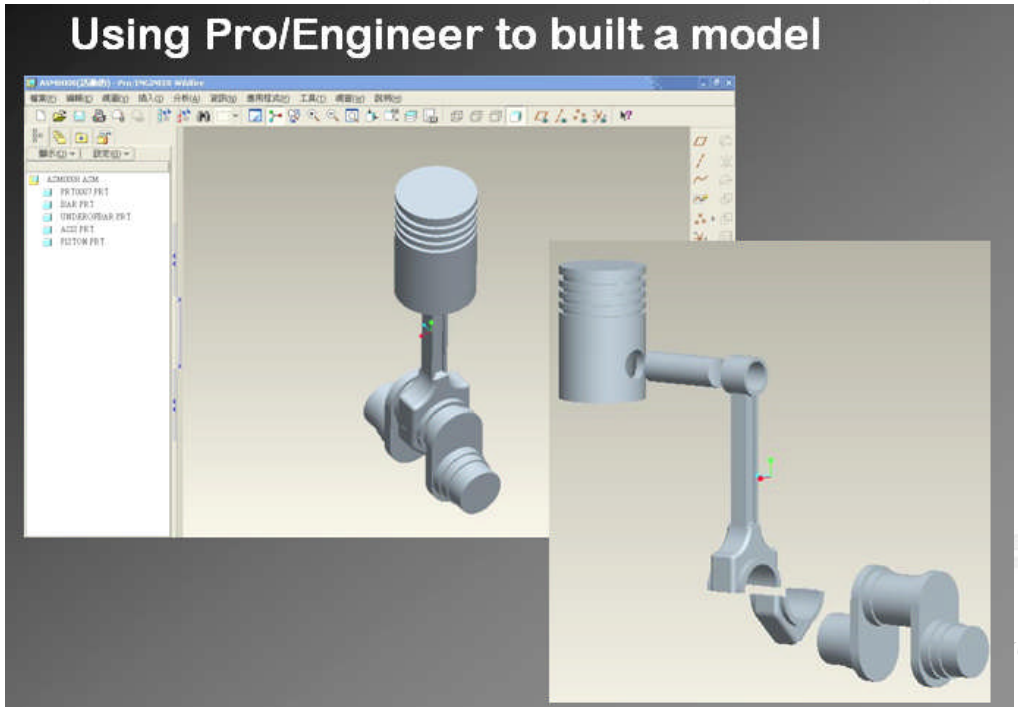
Know the piston



Dissection piston

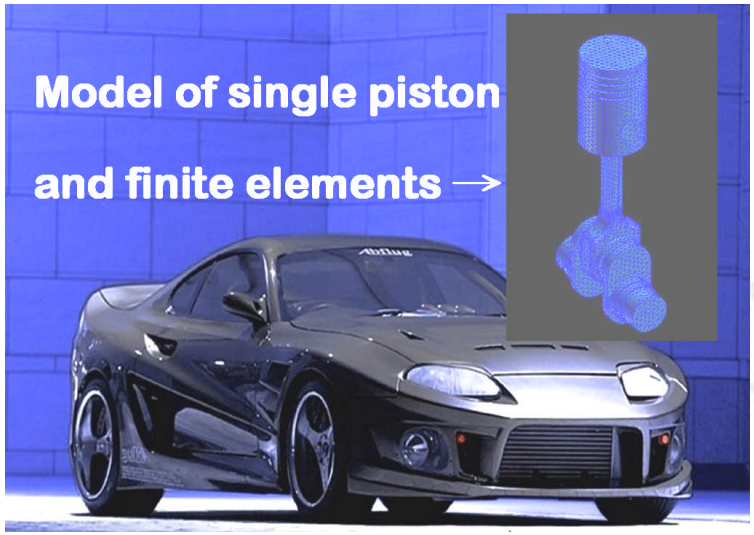


Using Pro/E built the model

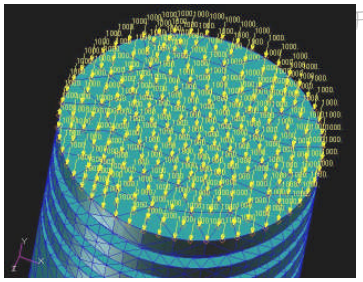


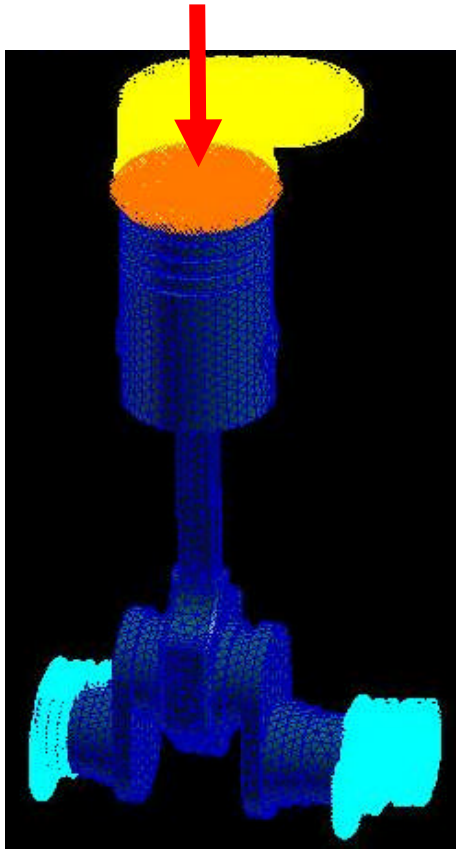
Put the model in MSC software SIM office

Using 3-D finite elements



Set up the boundary conditions (force =1000)





Boundary Conditions

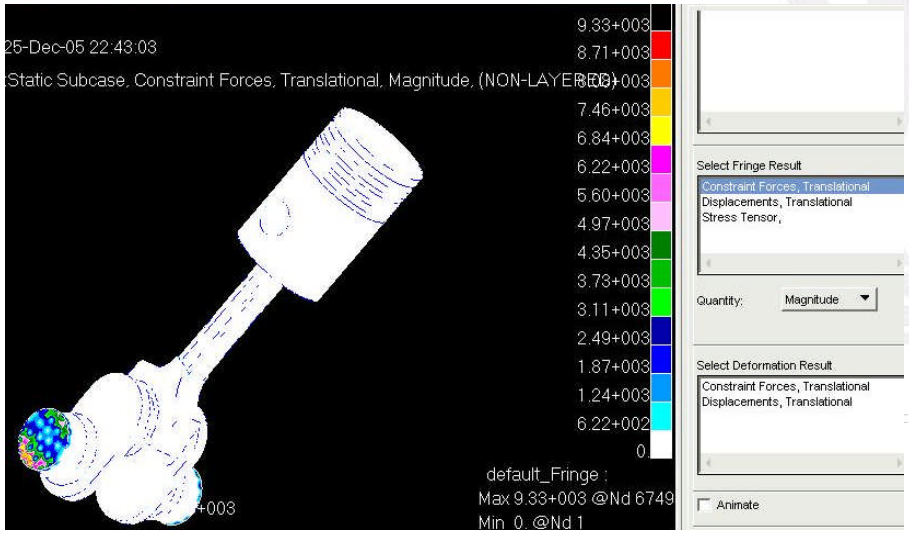
Theoretically, no boundary conditions are required. However, I do advise to apply at least one to a model, as it will help the solver overcome any potential problems due to rigid body motion (otherwise known as ill-conditioning). Just fix any point on the model with a zero displacement, and the model will always solve.

If you intend applying boundary conditions, note that the results of eigenvalue analyses are very sensitive to the way in which BC's are applied. Therefore, always try to replicate the physical boundary conditions as closely as possible. If you cannot achieve this for some reason, carry out a sensitivity analysis by modifying the BCs slightly & comparing the results with your previous output. This will give you an indication of the influence the BCs have on the results.

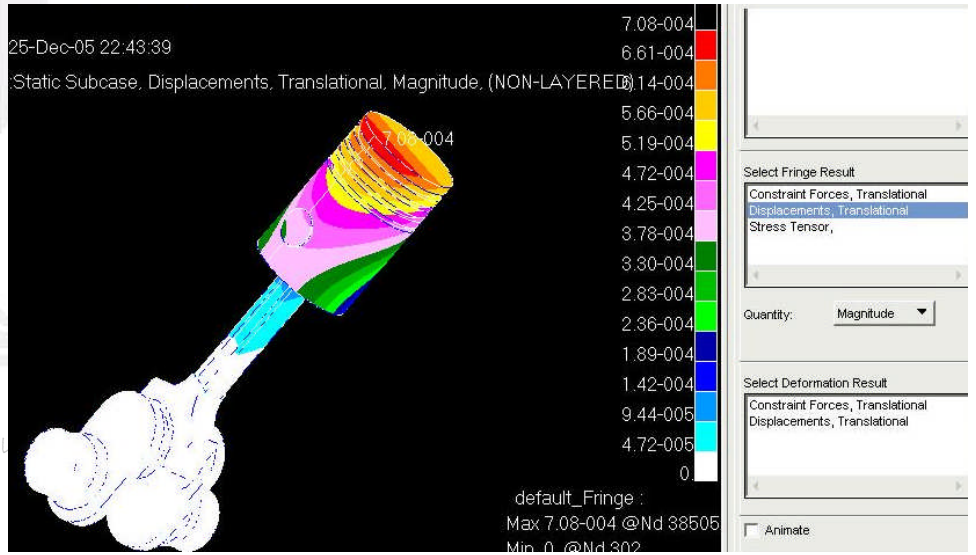
Analysis event

ALUMINUM (AL) 7075-T6

Constraint forces

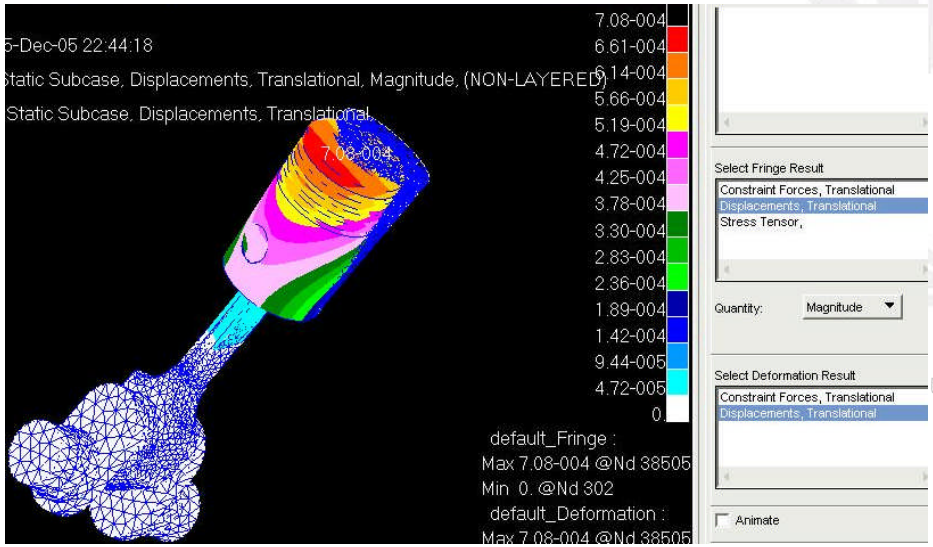


Displacement

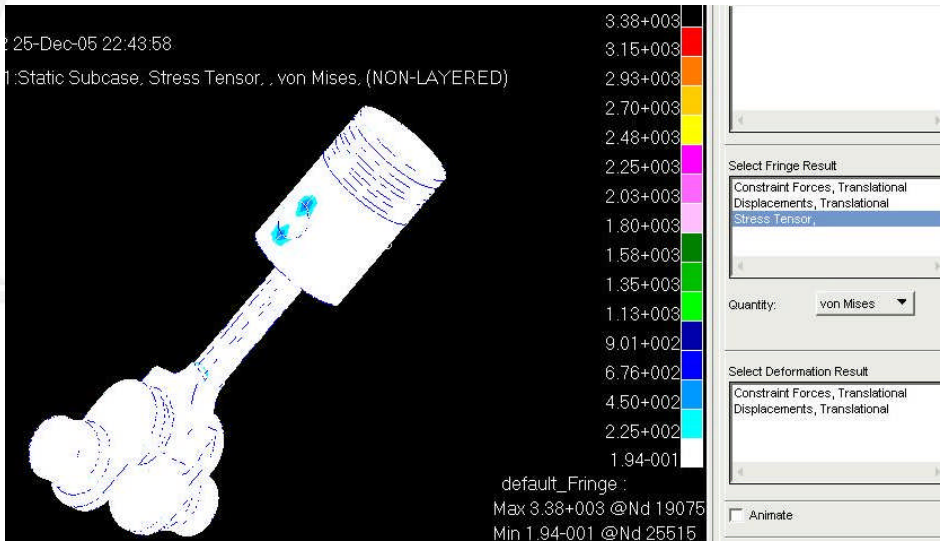


ALUMINUM (AL) 7075-T6

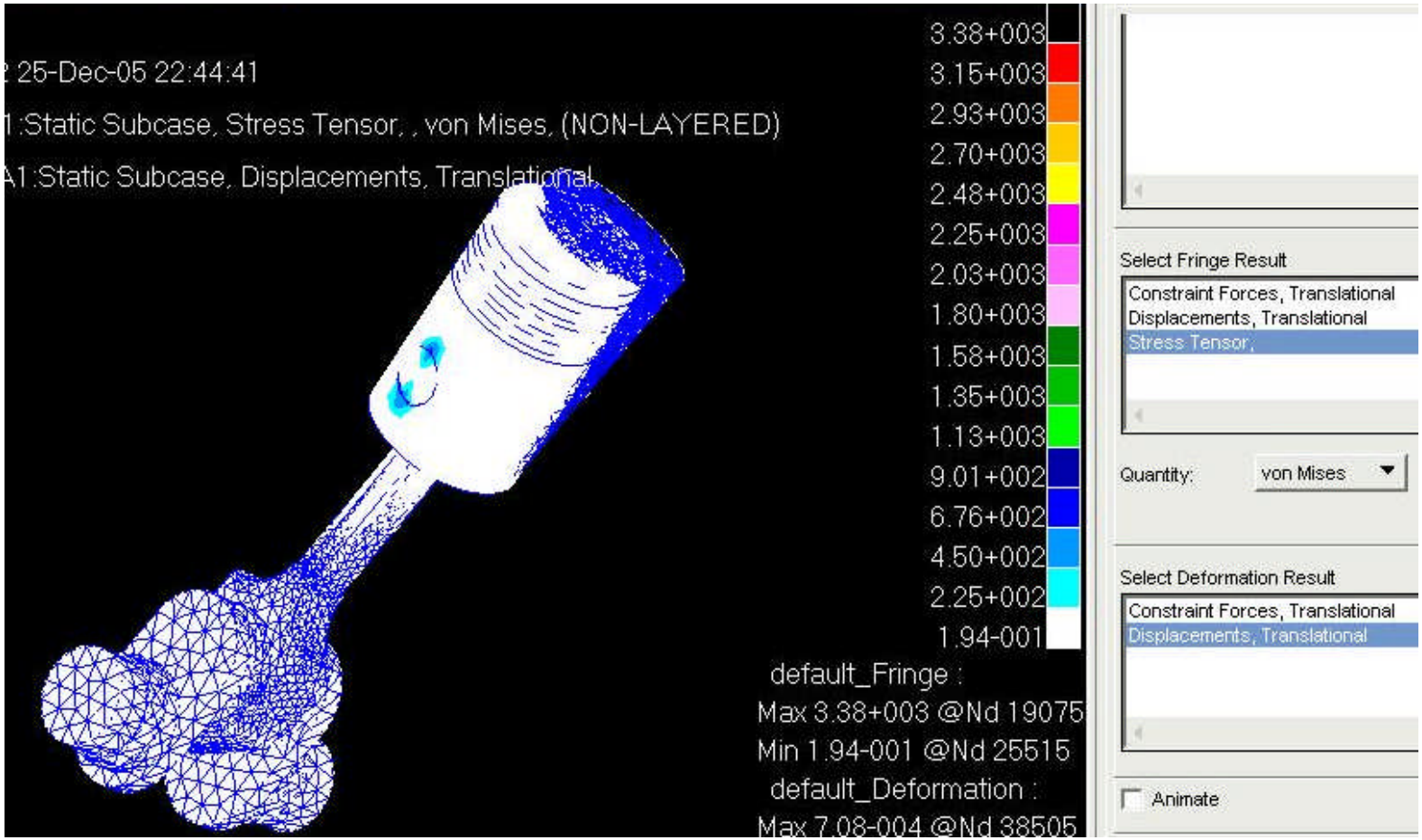
Displacement- Translational



Stress tensor

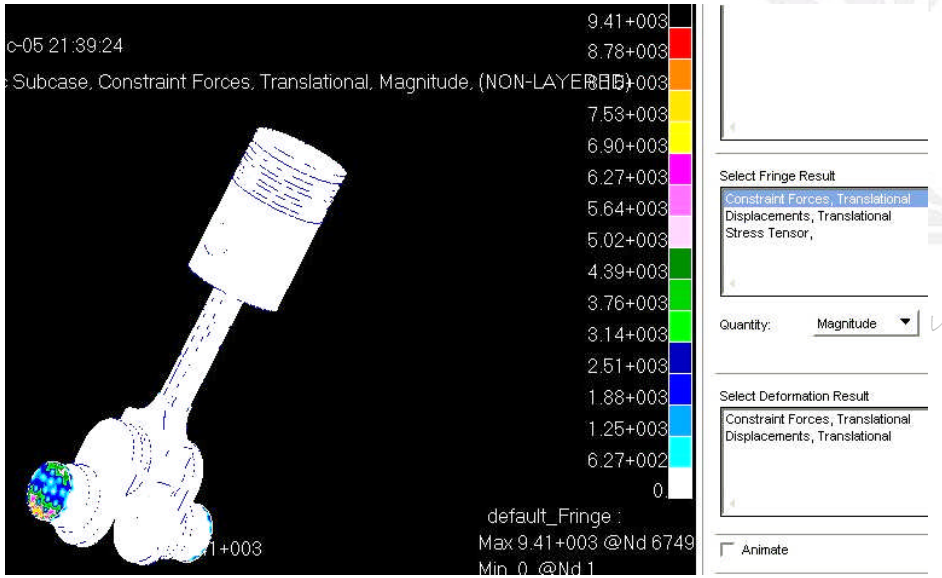


Stress tensor - displacement

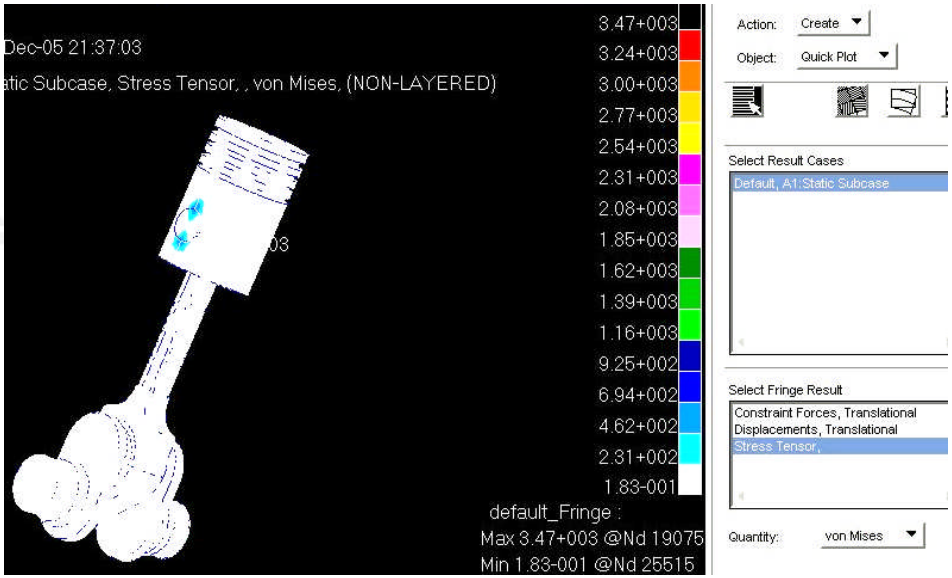


STEEL

Constraint forces

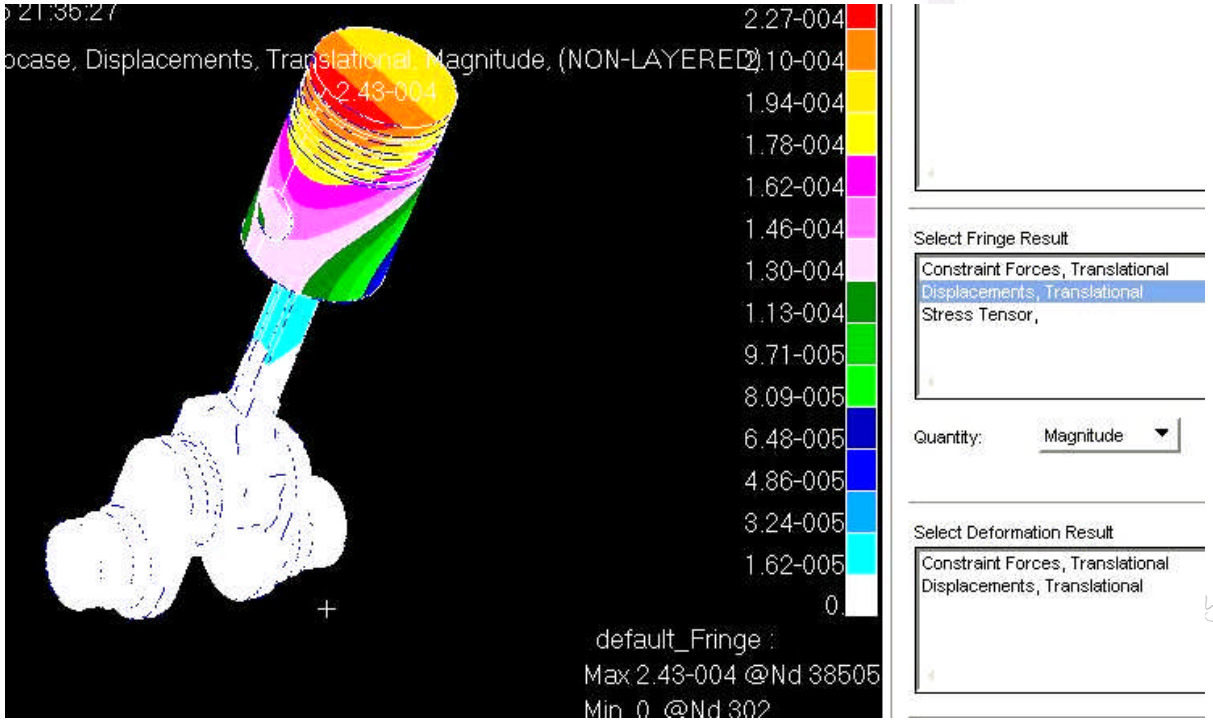


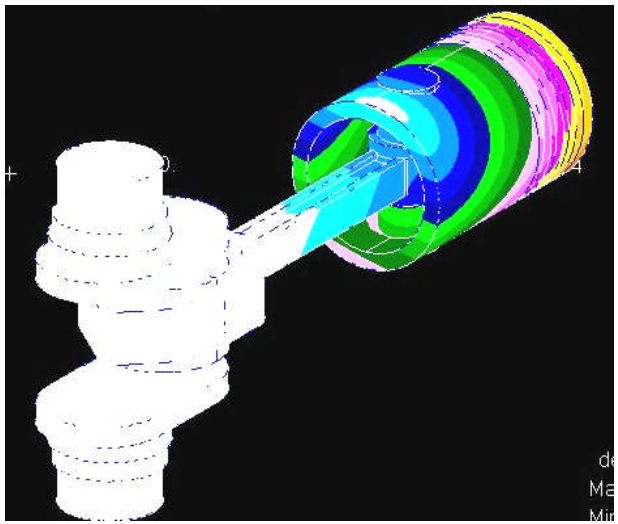
Stress tensor



STEEL

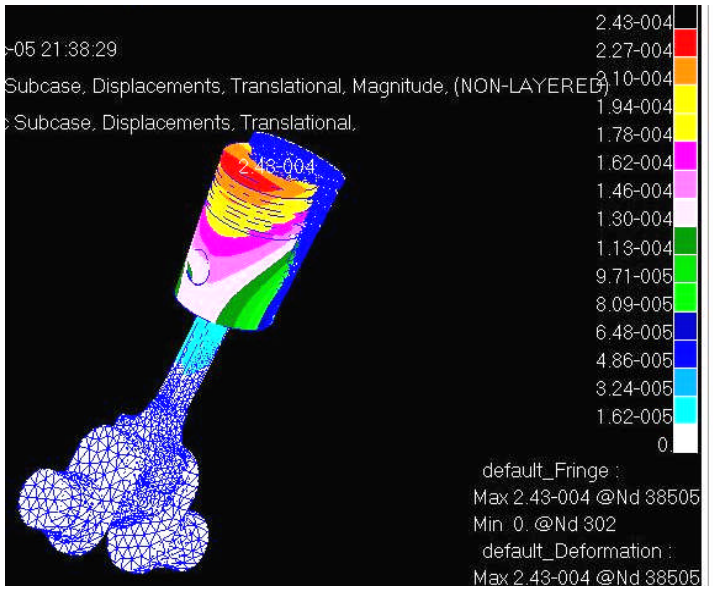
Displacement



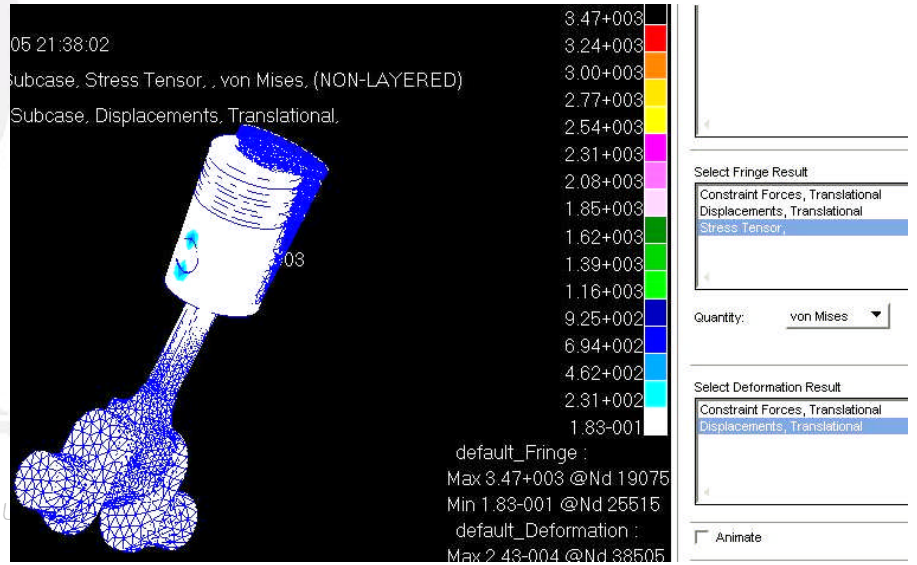


STEEL

Displacement- Translational

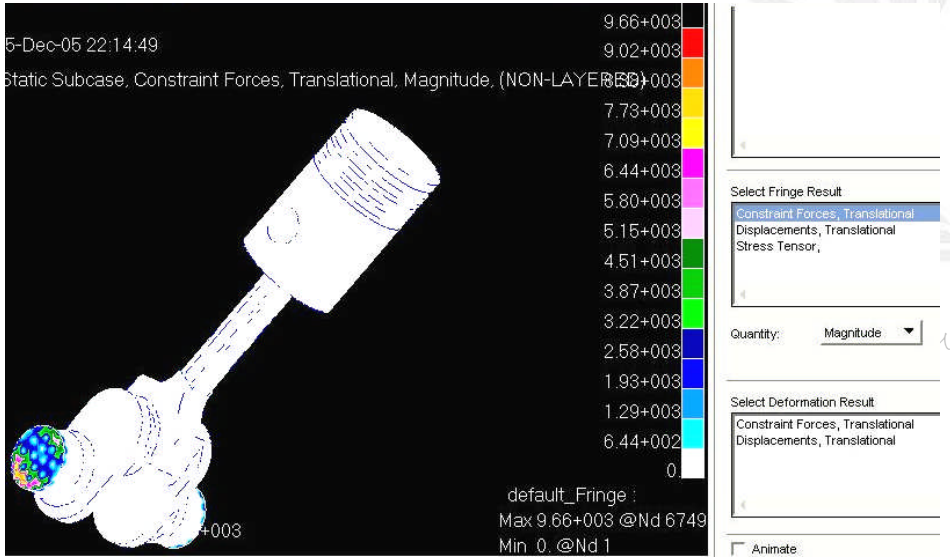


Stress tensor - displacement

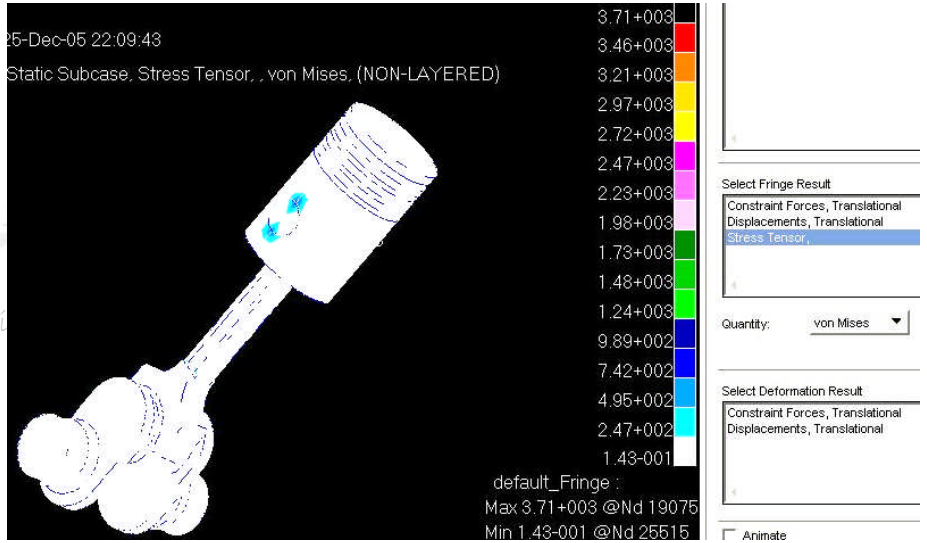


WOLFRAM (W)

Constraint forces

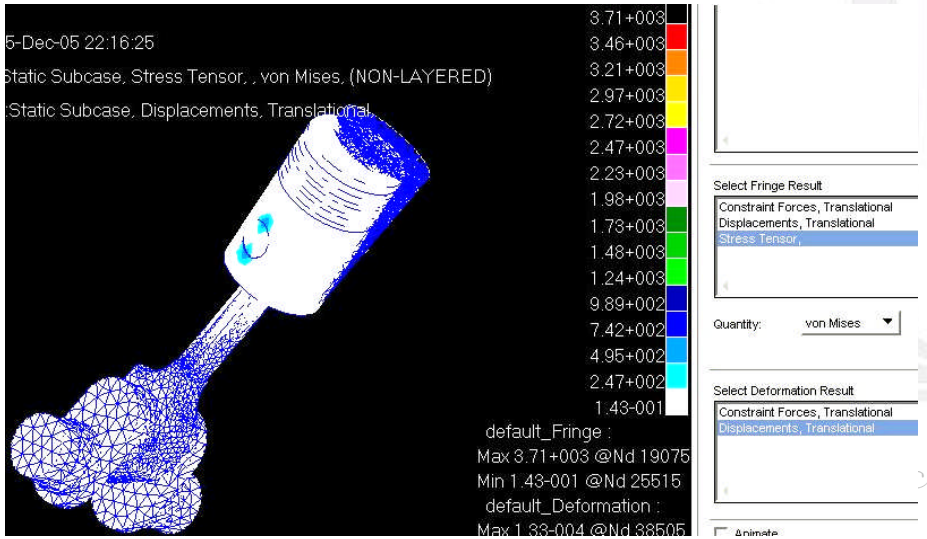


Stress tensor

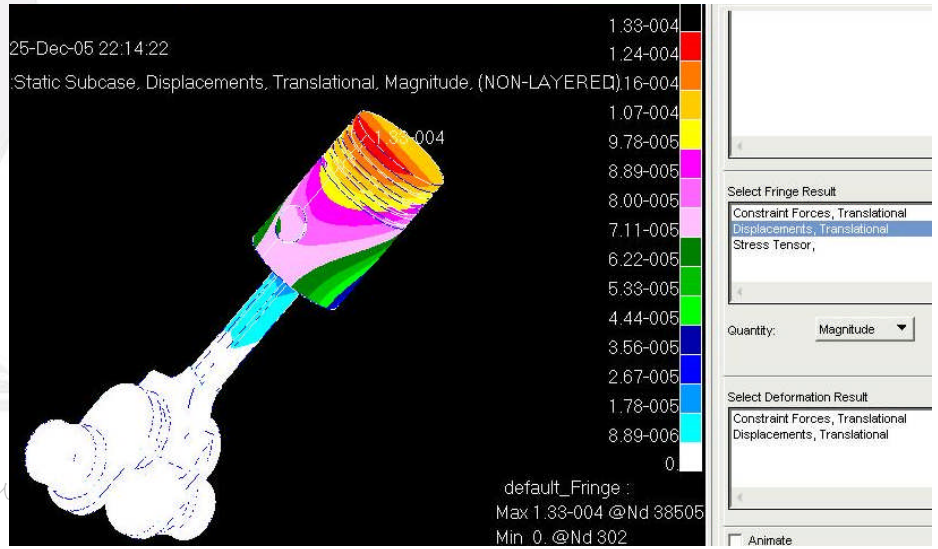


WOLFRAM (W)

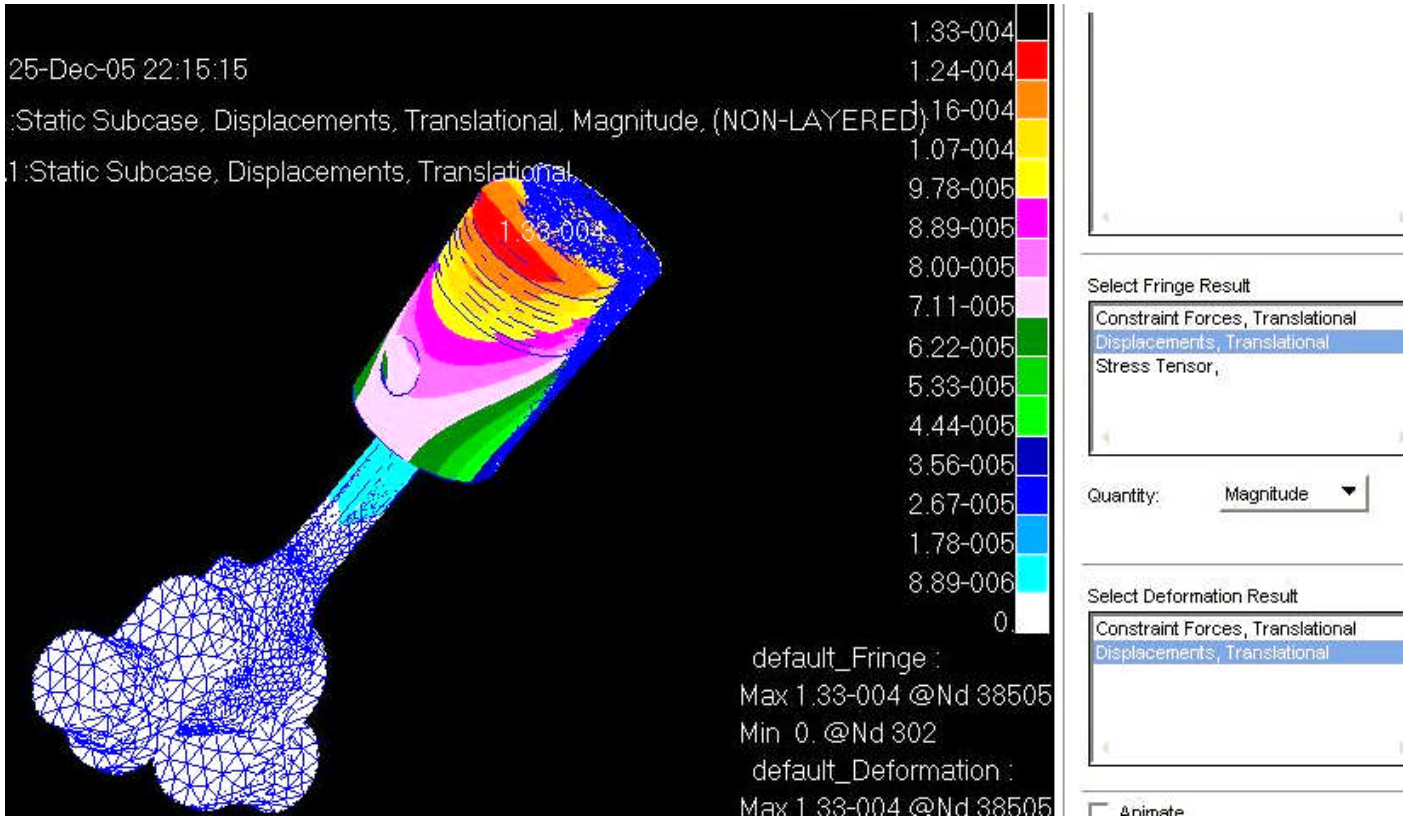
Stress tensor - displacement



Displacement

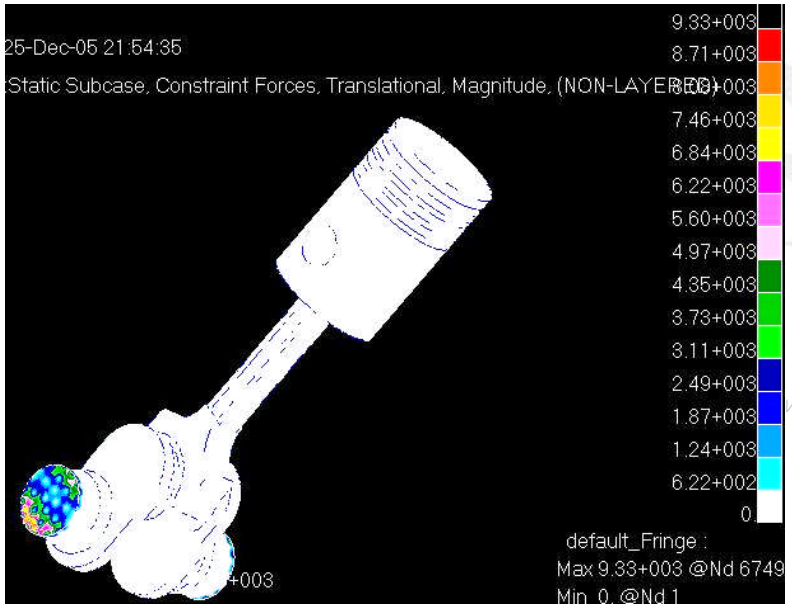


Displacement- Translational

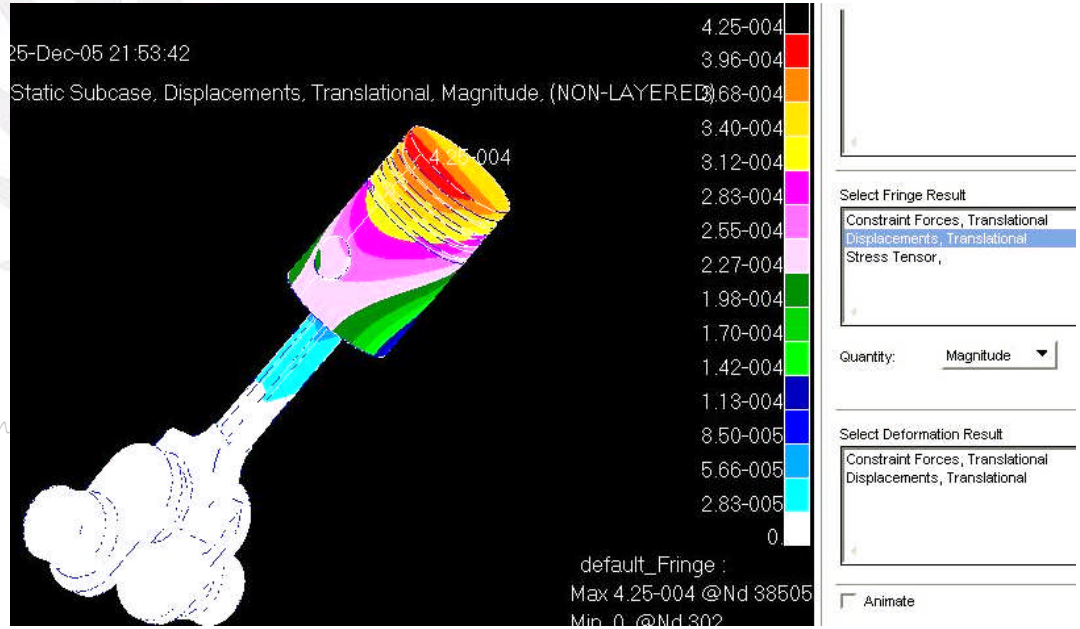


TITANIUM (Ti)

Constraint forces

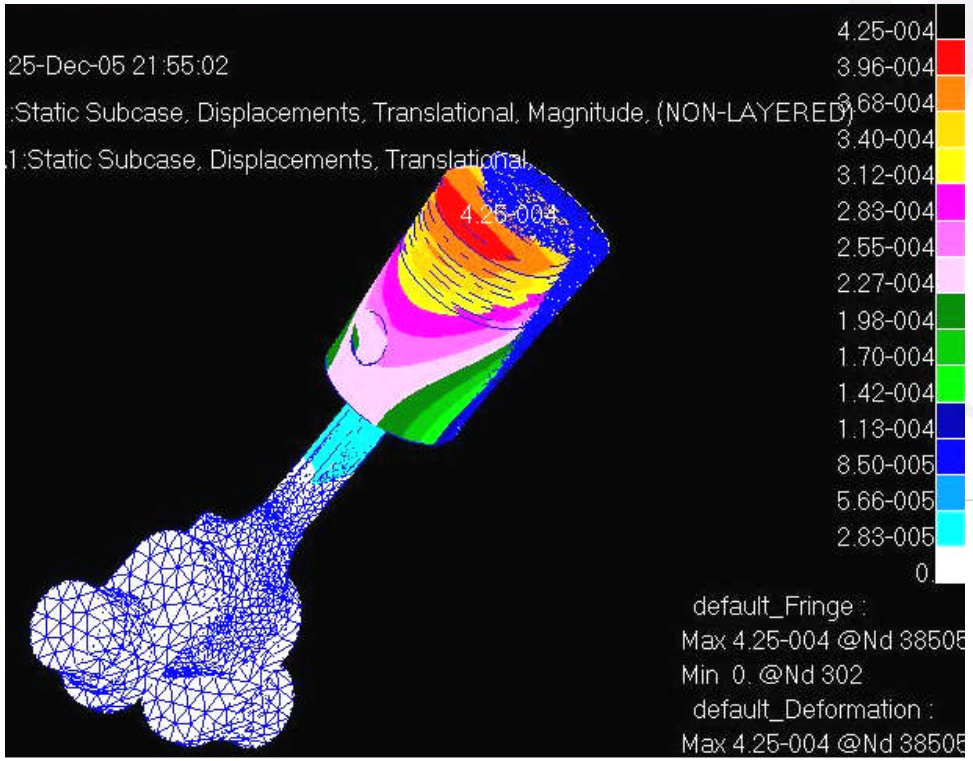


Displacement

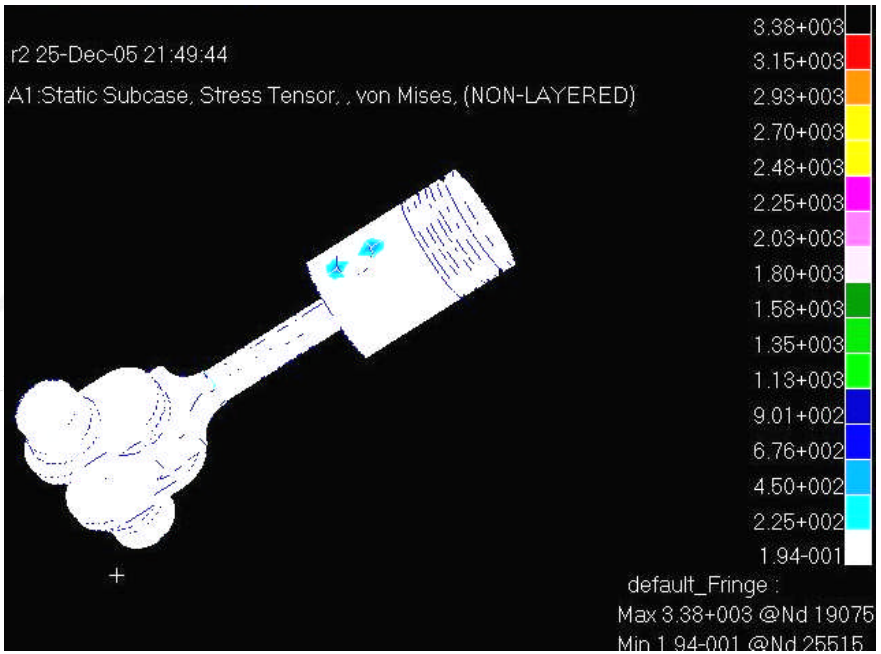


TITANIUM (Ti)

Displacement- Translational

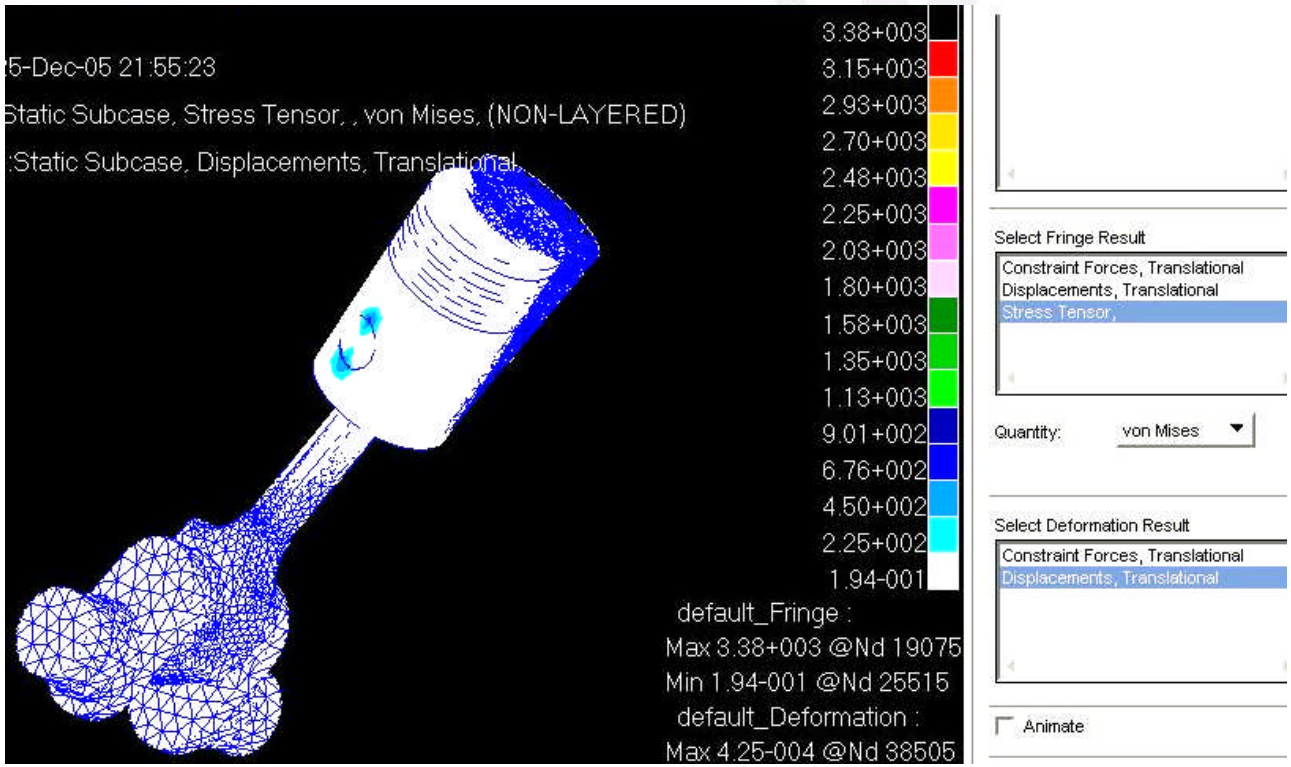


Stress tensor



TITANIUM (TI)

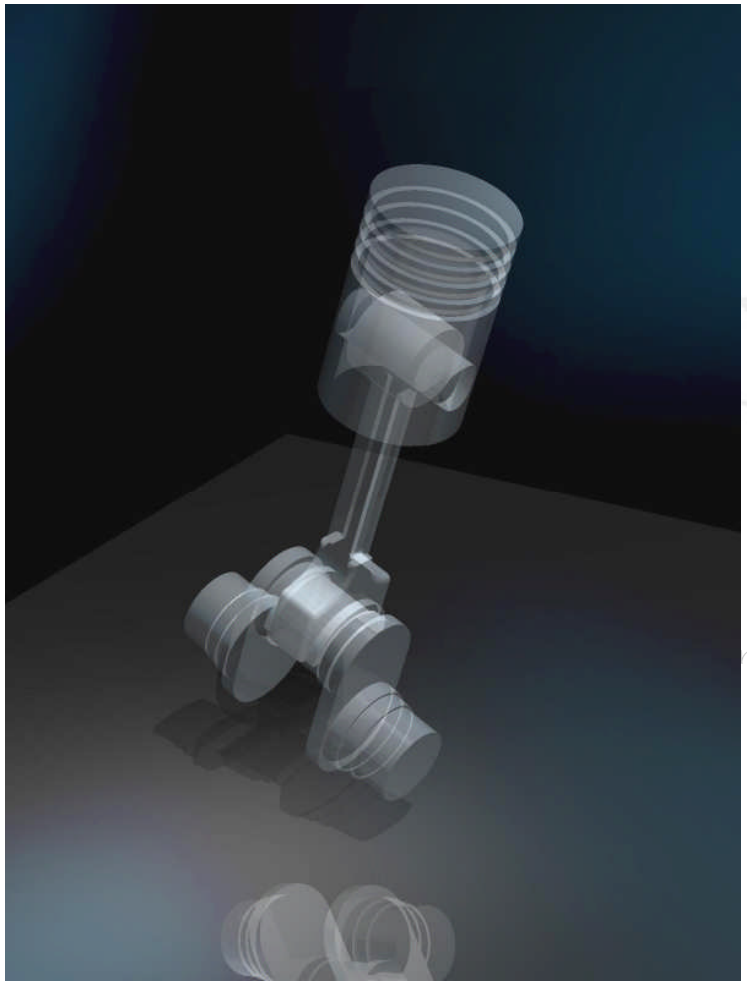
Stress tensor - displacement

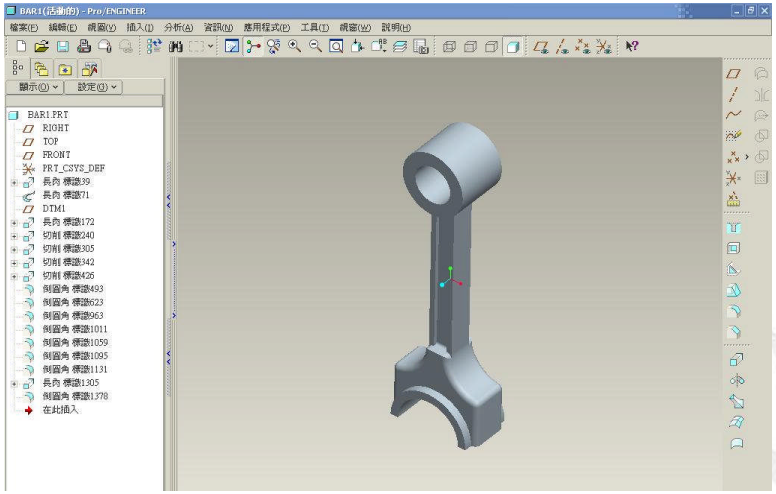


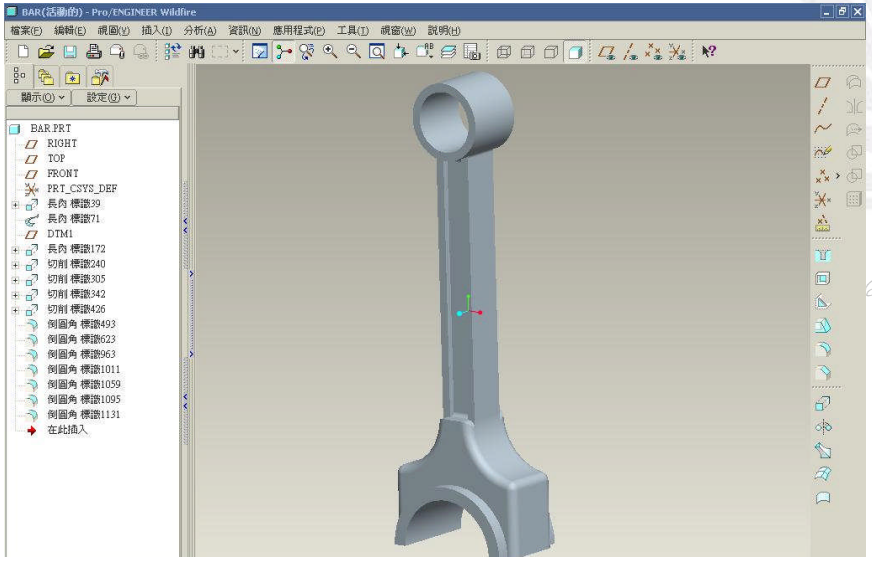
Amendment of new piston

We discover the stress concentration will happen at the intersection point of piston and the cam bar by this analysis results. If the material can't afford the stress concentration, which will be cracked at the intersection point, the stress concentration seems to be a circumstance we can't avoid. As intended engineers, we need to figure out how to decrease the deformation of the piston. At the results we manage to amend cam bars, and let the intersection part become wider and more depth. Finally, we use FEM to analysis it, and we find out the result really improves a lot.

Built a new model at Pro/Engineer software





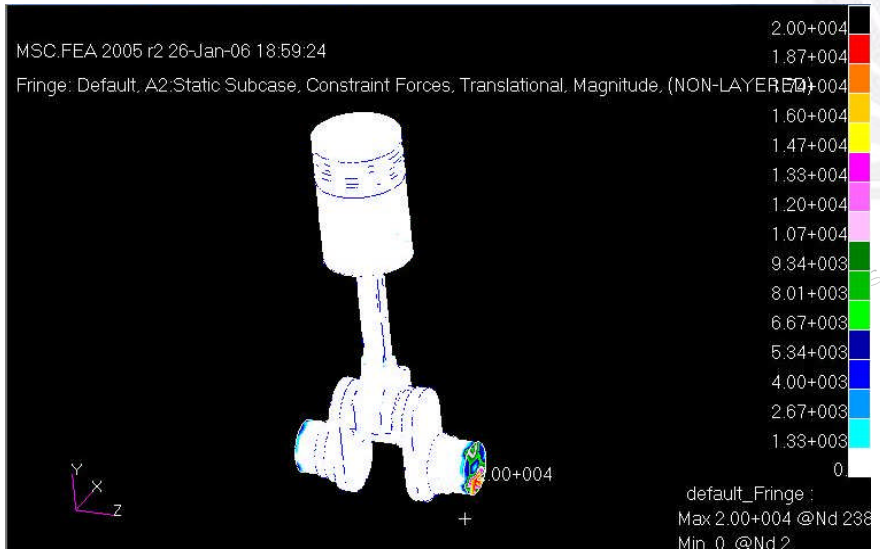


As compared with used

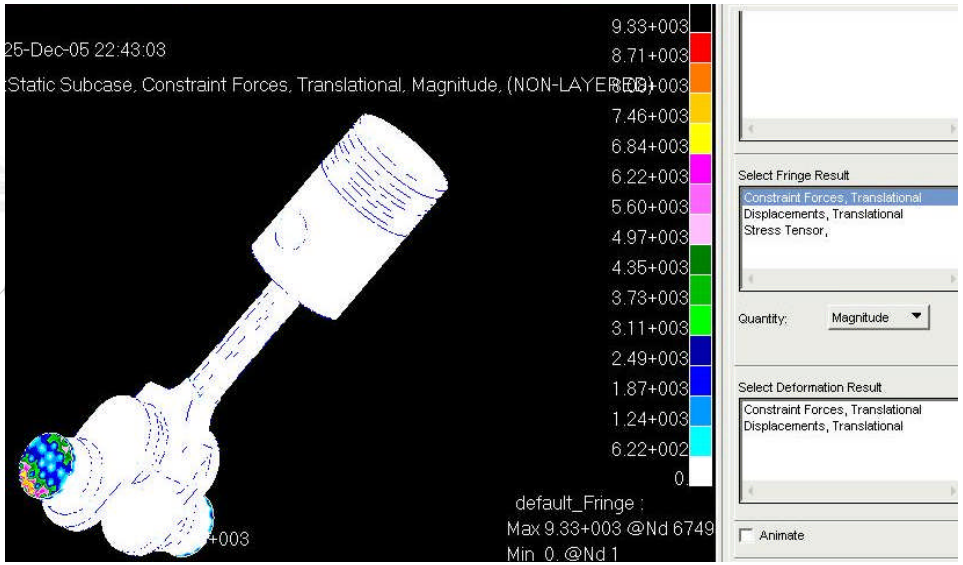
ALUMINUM (AL) 7075-T6

Constraint forces

Amendment

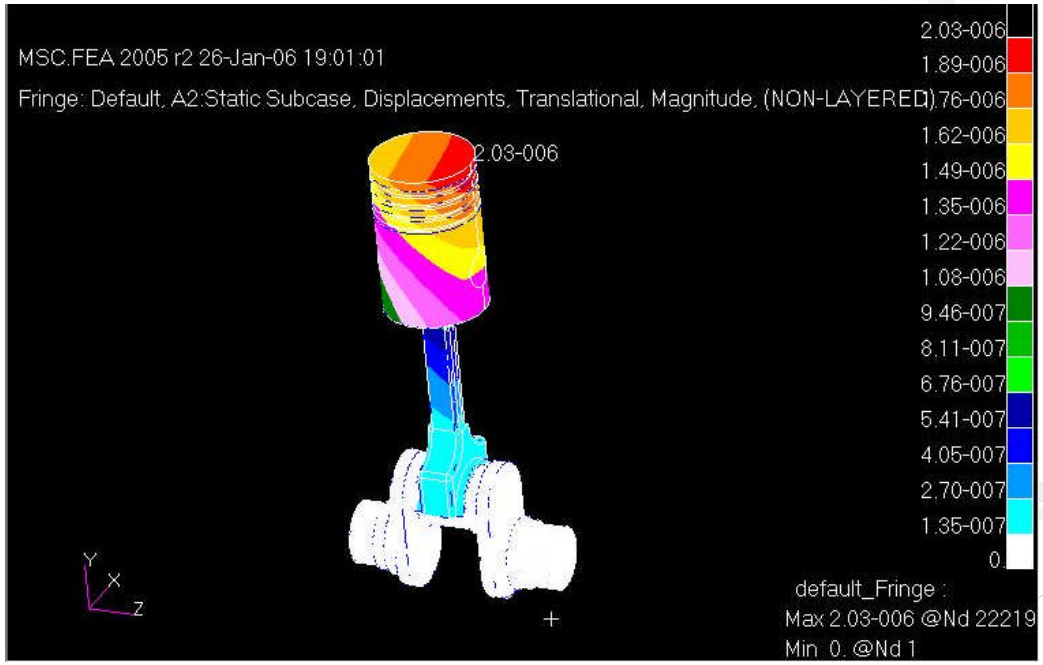


Used

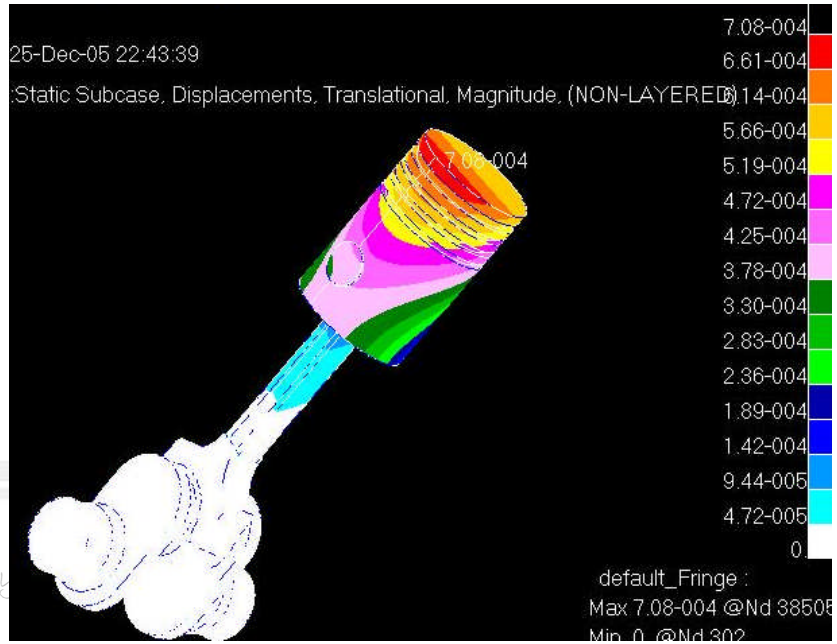


Displacement

Amendment

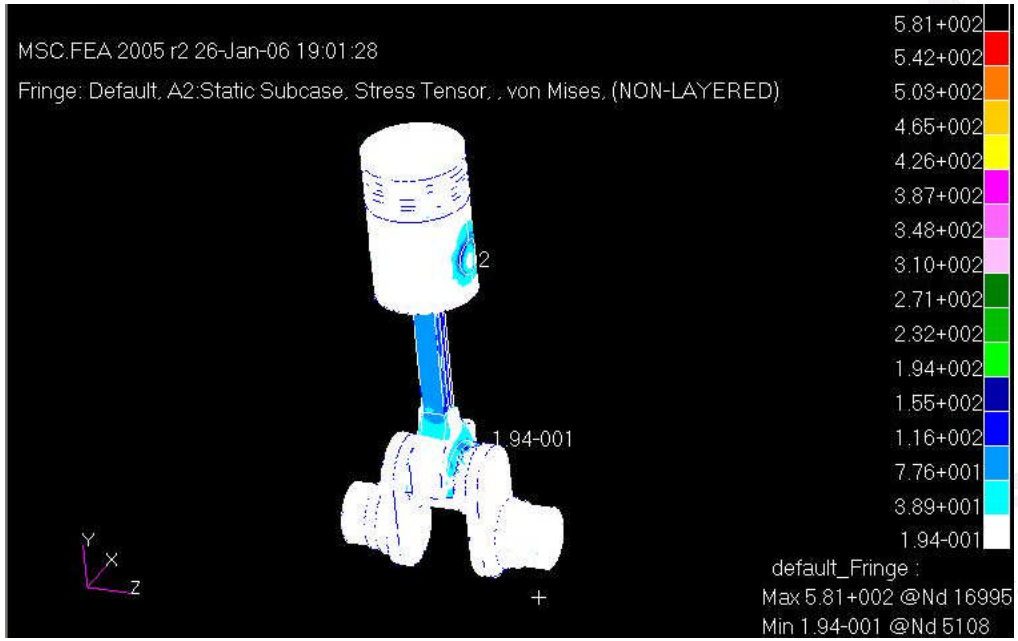


Used

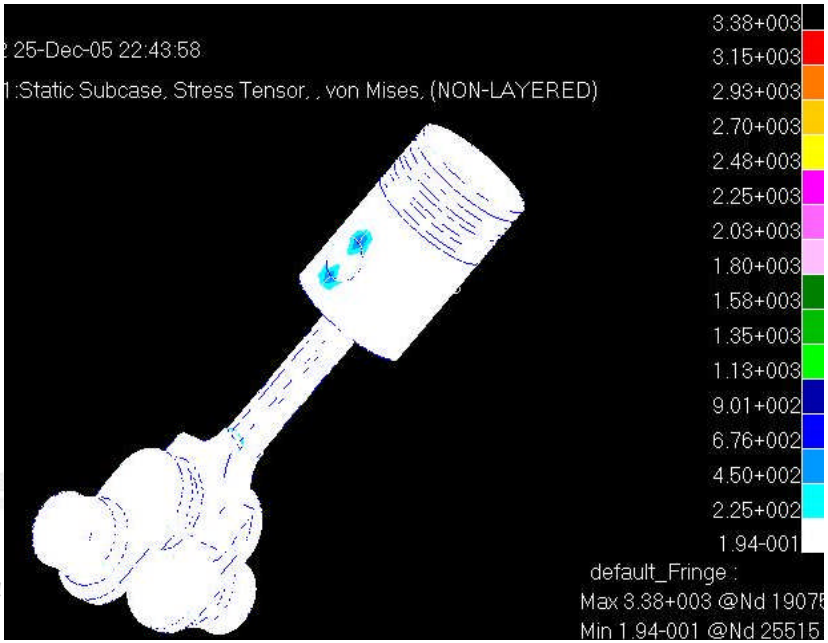


Stress tensor

Amendment



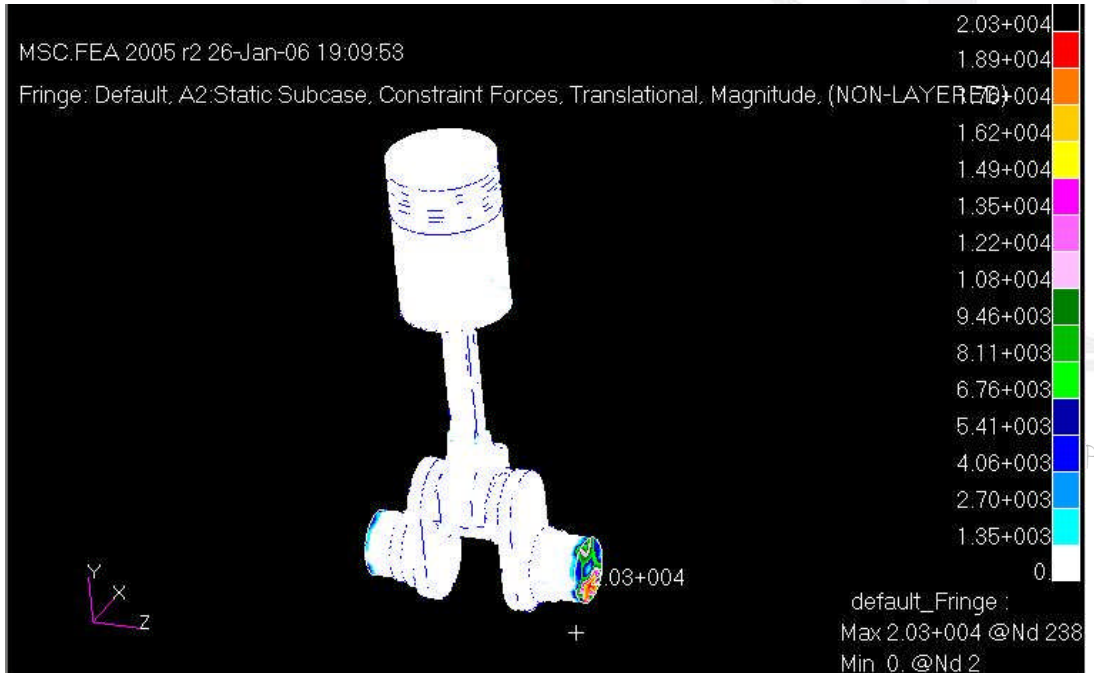
Used



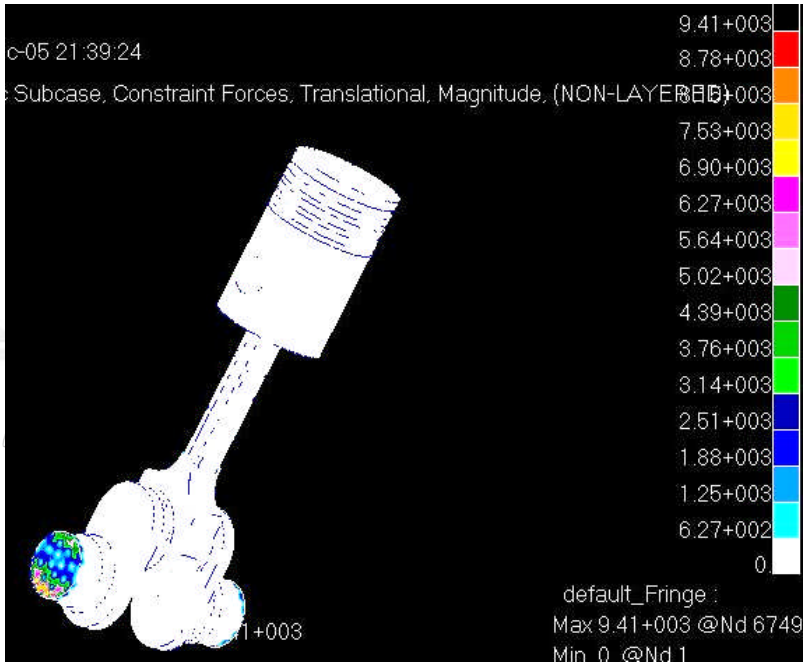
STEEL

Constraint forces

Amendment

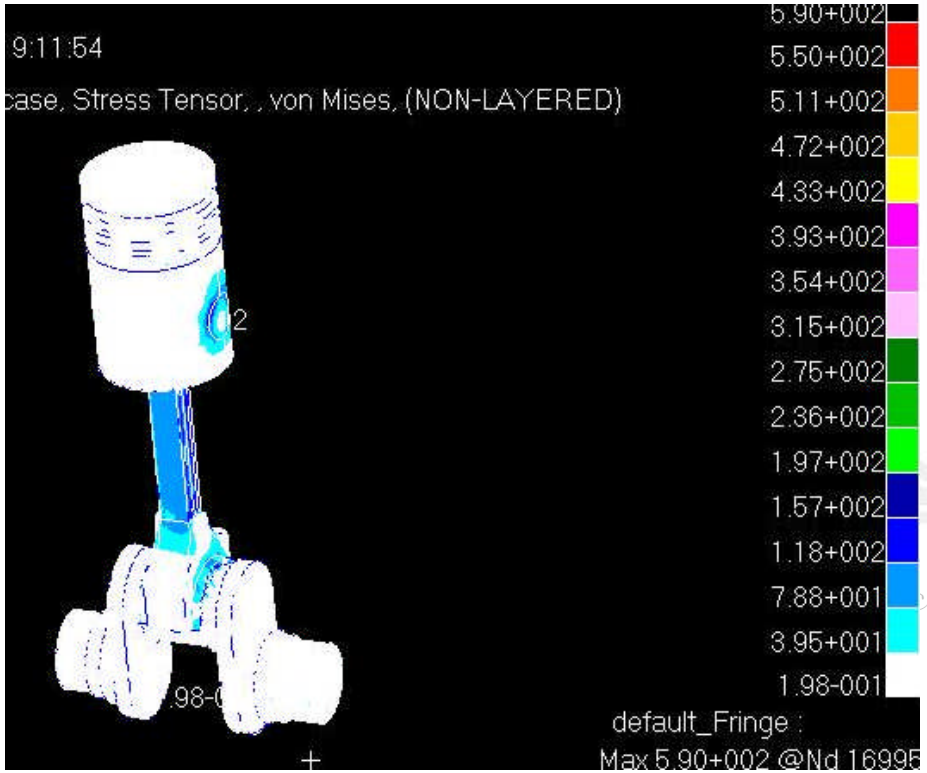


Used

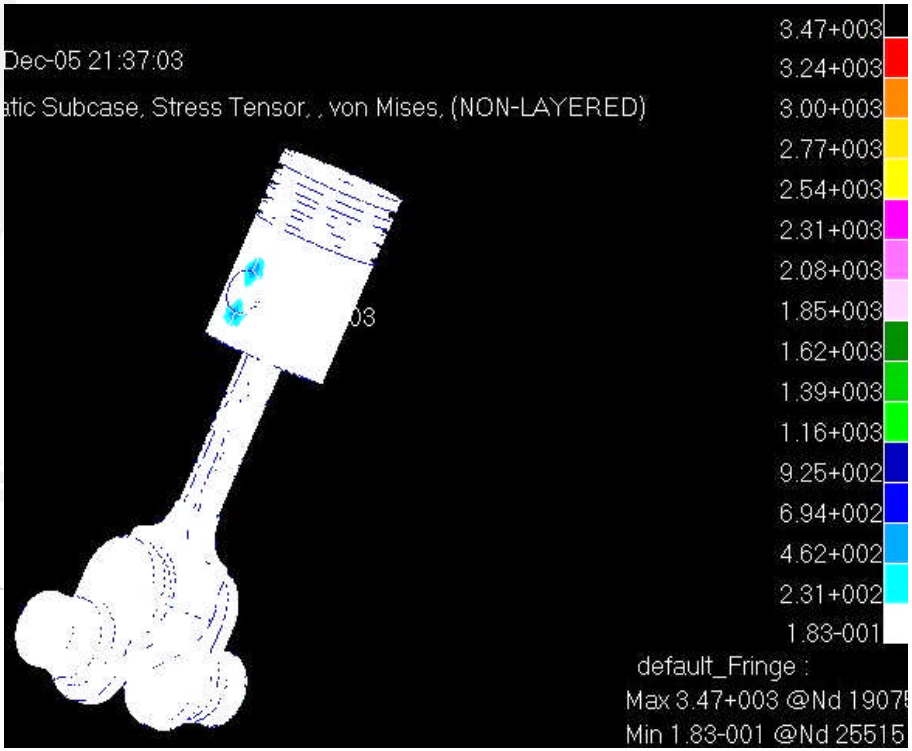


Stress tensor

Amendment

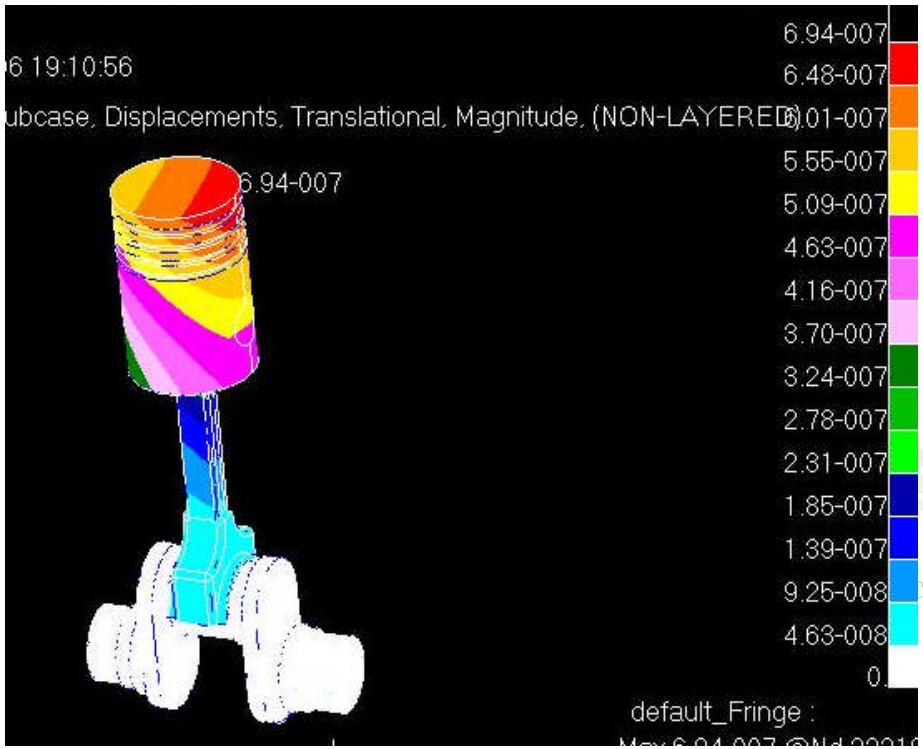


Used

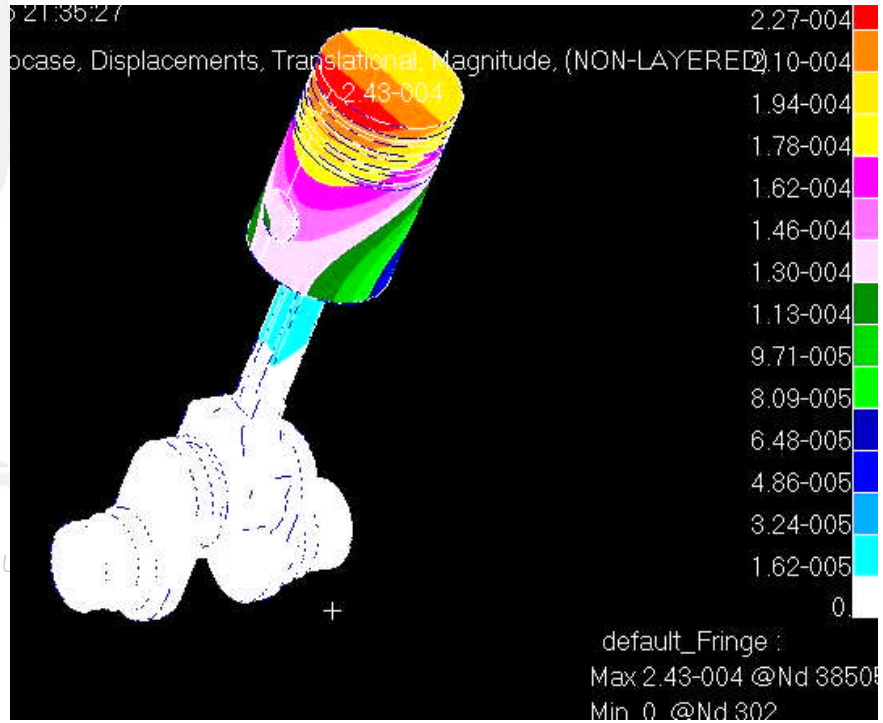


Displacement

Amendment



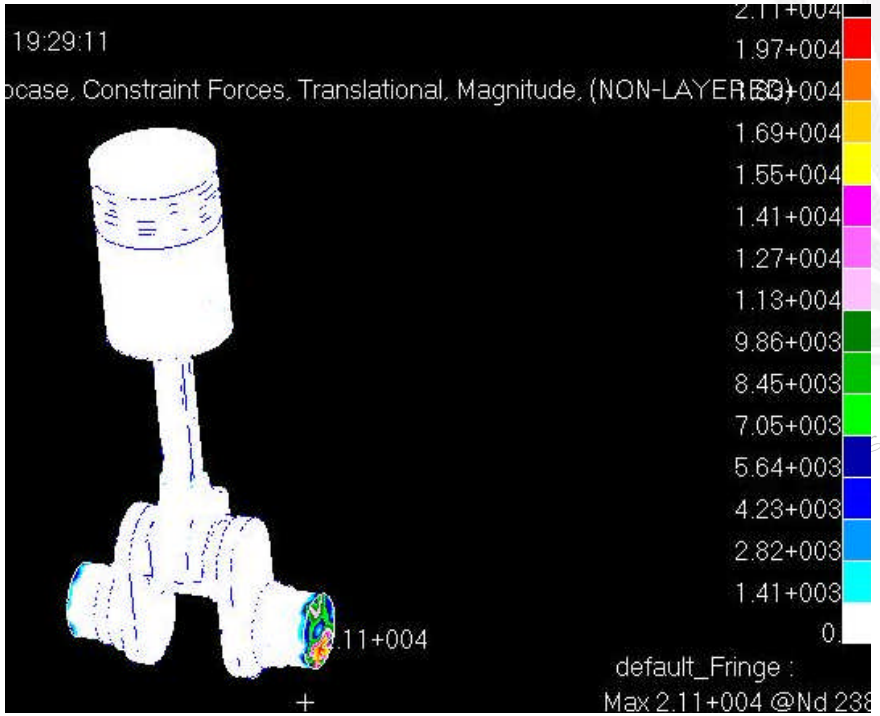
Used



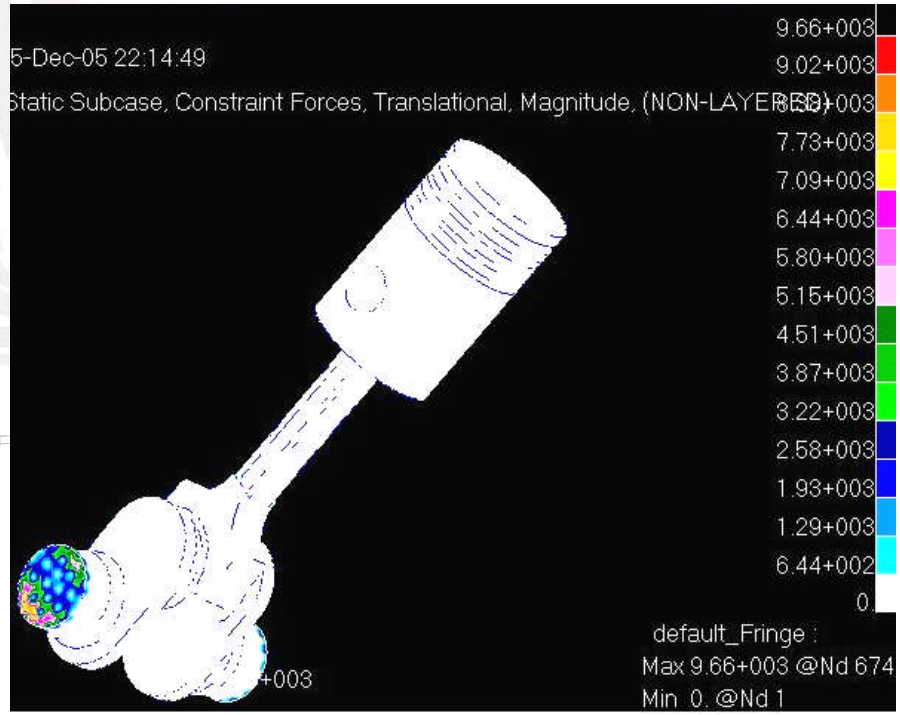
WOLFRAM (W)

Constraint forces

Amendment

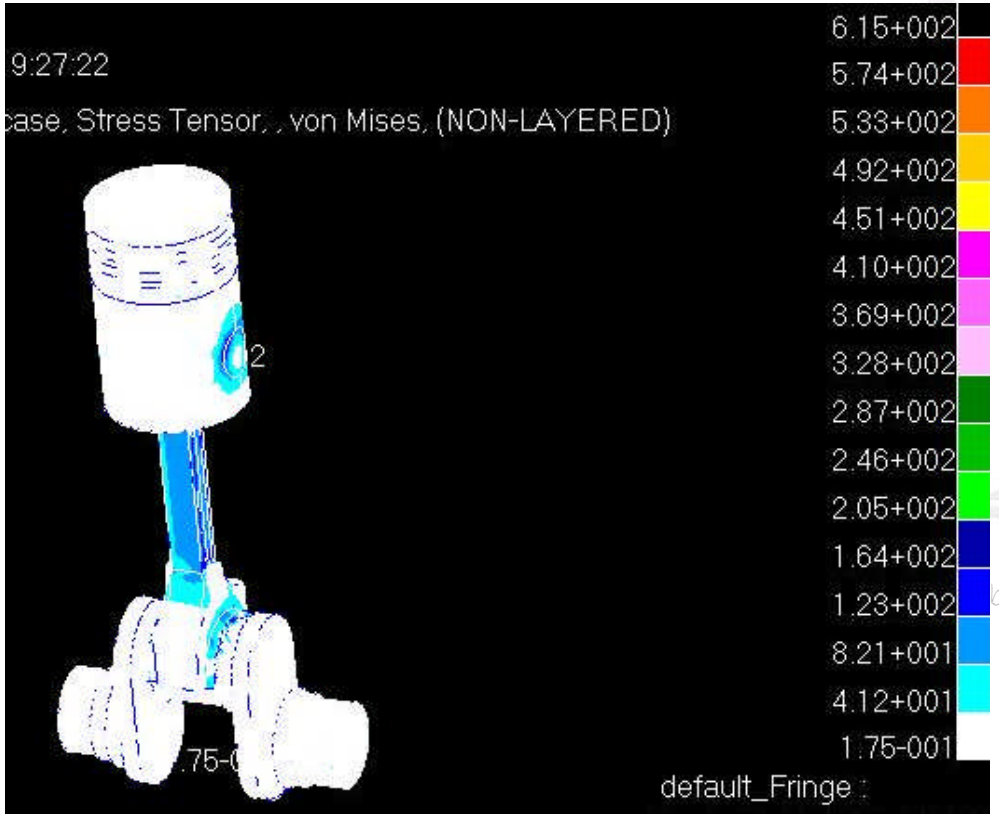


Used

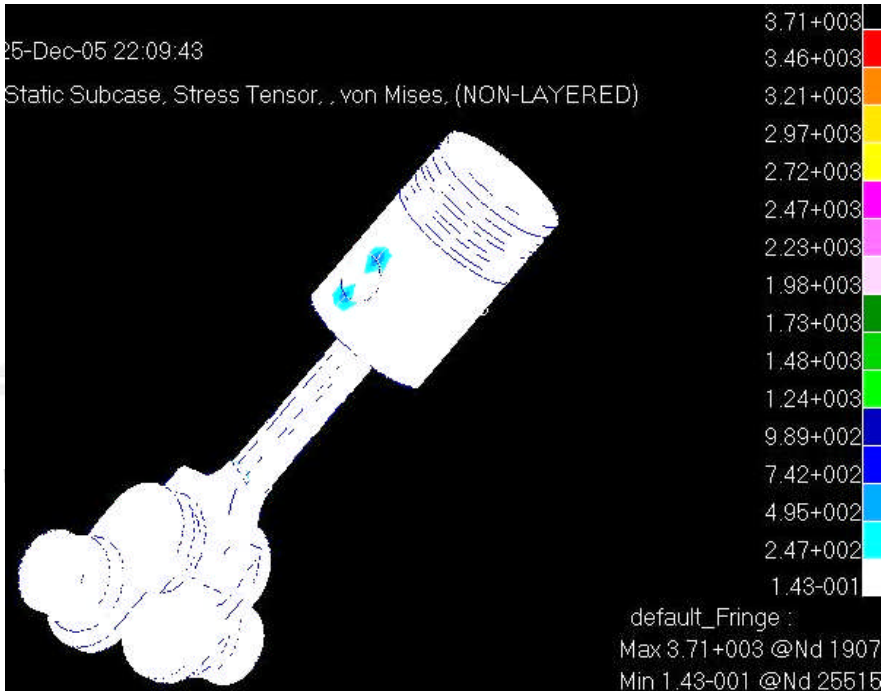


Stress tensor

Amendment

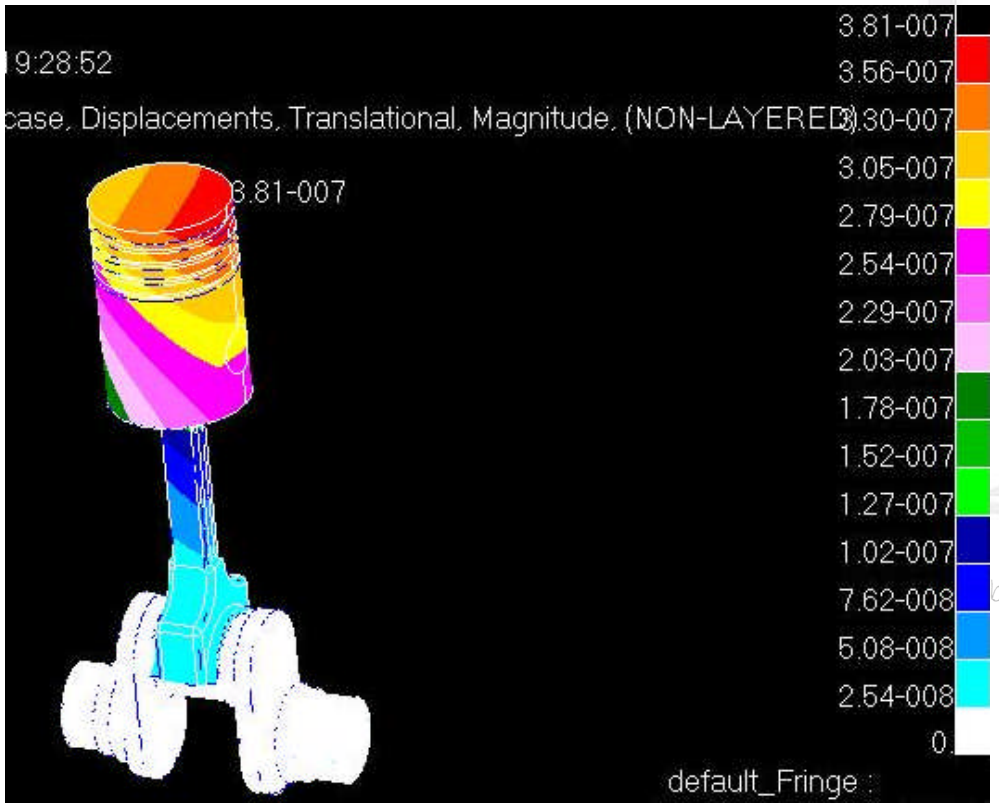


Used

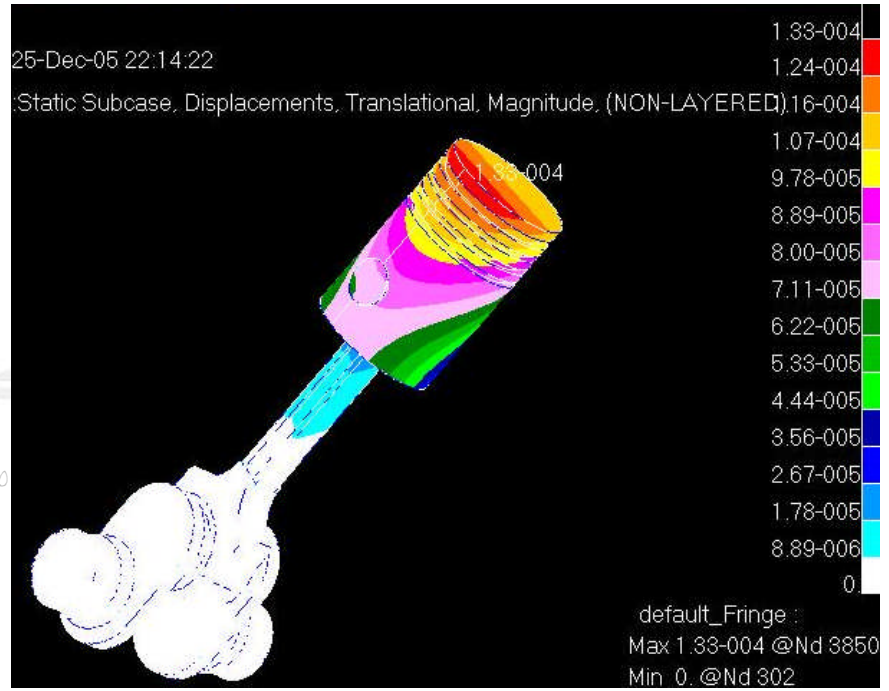


Displacement

Amendment



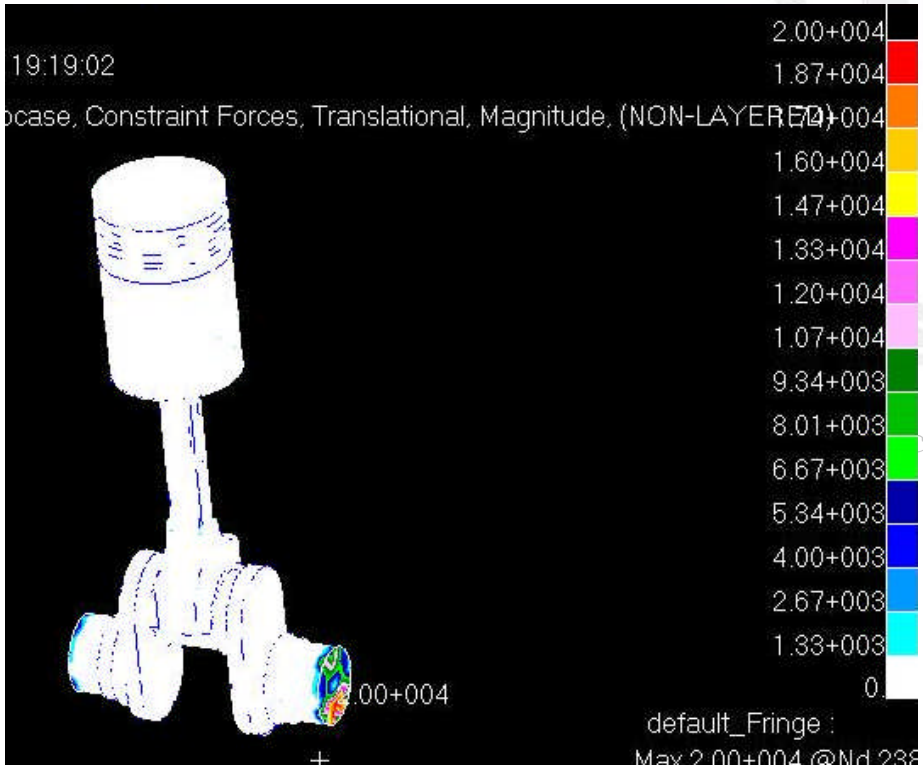
Used



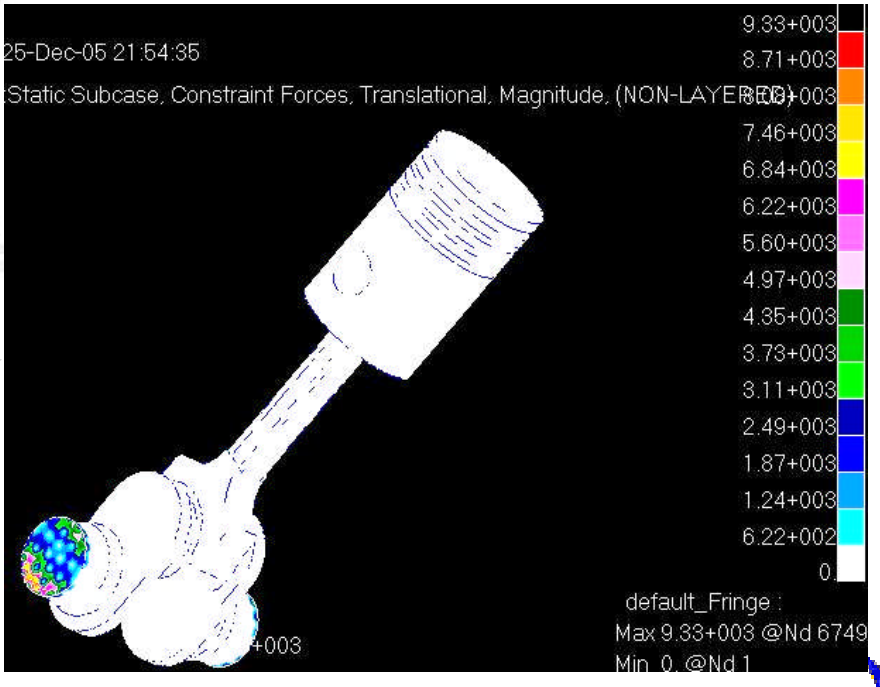
Titanium (Ti)

Constraint forces

Amendment

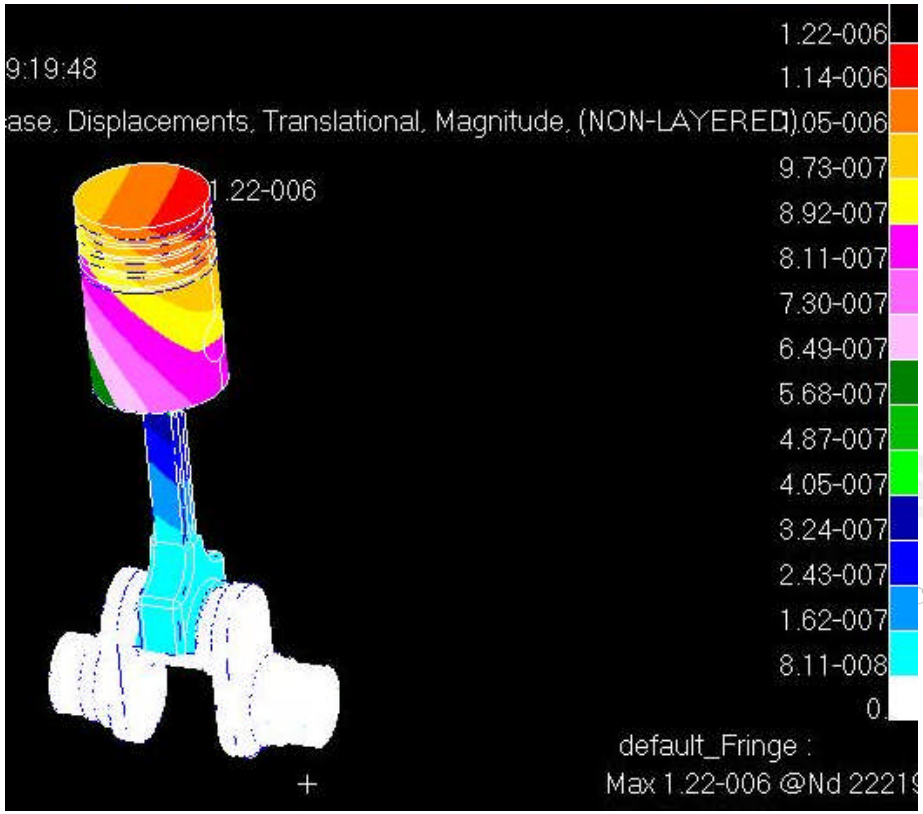


Used

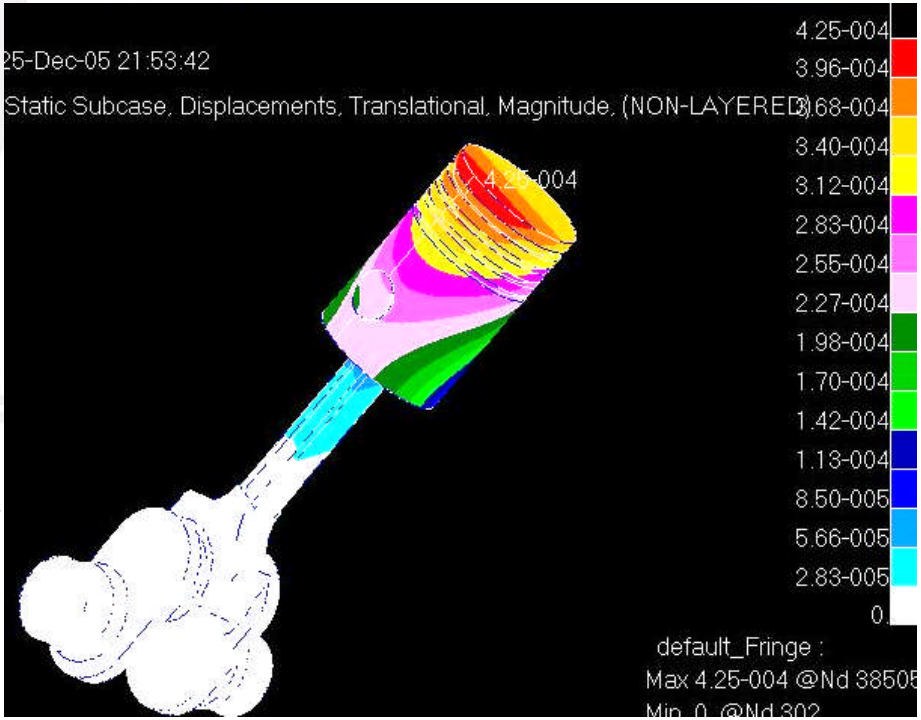


Displacement

Amendment

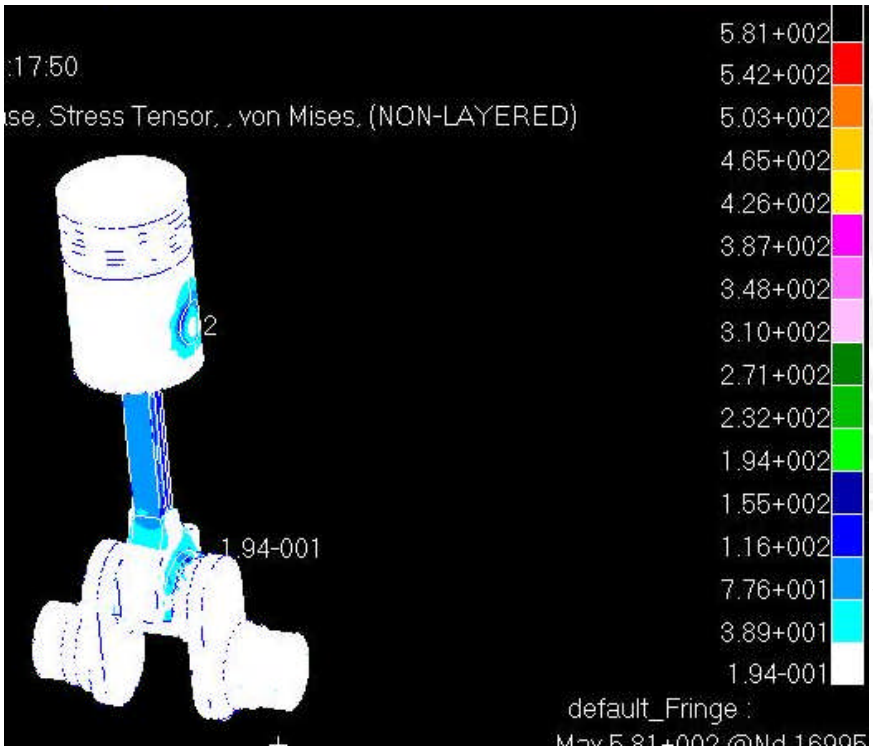


Used

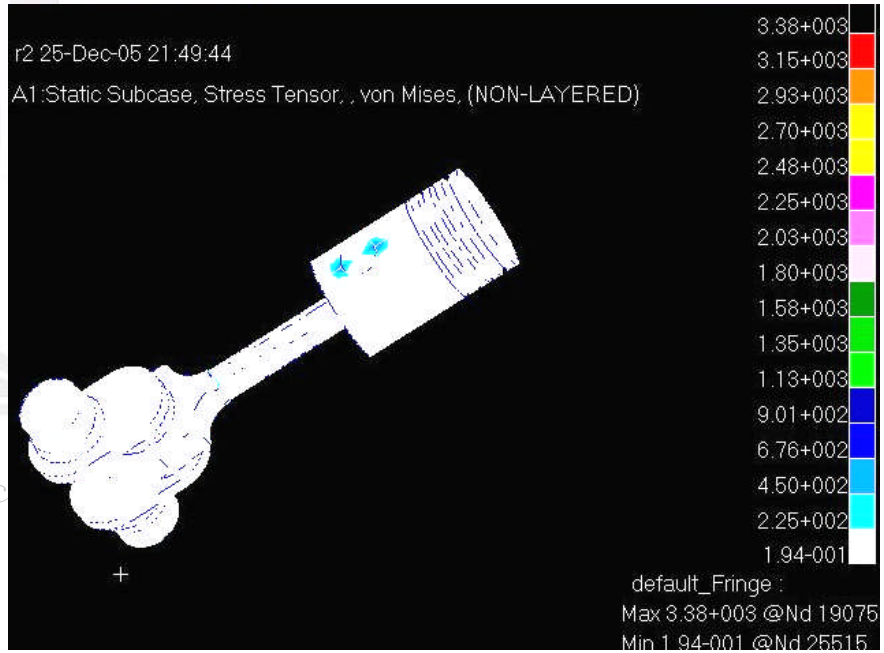


Stress tensor

Amendment



Used



Sentiments

時凡浩

The reason that I choose this piston to analysis the final project is because I'm very interesting in it. As we all know the power of the car is because the piston in car's engine keeps on blowing and the air in it is expansion once and once. Therefore, material plays a very important rule in the piston. If the engineer uses wrong materials, the result will be very horrible. By this class of the analysis software- Nastran- uses the finite element method to analysis it's stress and strain to judge the material, which is good enough to the piston.

盧亭君

Using the skills we had learned by try more challenge though the skill we have learned this semester and find out how to use on the thing we interesting. In our team we had discuss many time, from how do we choose our topic and then how do we built the model ... the questions are always one comes by another. The beginning step building the finite elements we have try it many time, even have to let the computer to run more then twenty minuets, we often think maybe is our mistake, until the answer were show out. As the day we finish maybe we still didn't know all about this FEA software, but as we try more we can do it better at the next time!

Reference

MSC Software Inc, MSC.Patran (PAT301), MSC Software Inc
MSC Software Inc, MSC.FEA, MSC Software Inc
MSC Software Inc.n, MSC.NASTRAN (NAS101), MSC Software Inc
S. Moaveni, Finite Element Analysis -Theory and Application with ANSYS, Prentice Halln
J.Y. Li, E. Lee, S. Hsieh, and D. X. Xiang, Tutorial Manual, MSC Software Inc
J.Y. Li, E. Lee, S. Hsieh, and D. X. Xiang, Example Problem Manual, MSC Software Inc
林清安(民 92), PRO/Engineer 零件設計-基礎篇（上）Wildfire，2003, 臺北市 知城數位科技股份有限公司
林清安(民 92), PRO/Engineer 零件設計-基礎篇（下）Wildfire，2003, 臺北市 知城數位科技股份有限公司
劉紀嘉、蔣德元、詹世貝、無繡美、邱羿瑄、黃穎豐…2003[民 92], Pro/Engineer Wildfire 綜合應用，2004, 臺北縣五股鄉 高立圖書有限公司 新科技總經銷