3-D CAD Modeling and Analysis of Aircraft Wing Using CATIA® Software And Its Comparison With ANSYS® Software

HASSAN NASEEM KHAN, M. WAHAB USAMA, RIAZ AHMAD, IQBAL RASOOL
Aerospace Engineering Department
College of Aeronautical Engineering
National University of Sciences and Technology, Islamabad
PAKISTAN
hassannaseemkhan@gmail.com;mwusama@yahoo.com;riazcae@yahoo.com;irmemon@hotmail.com

Abstract: -CAD model of parts play pivotal role in the design and development phases of an aircraft. Its uses are multiple, starting from FEM and CFD analysis at early design phase to all subsequent developments processes. CAD model and FEM analysis of an aircraft wing modelled in CATIA software is presented in this paper. Complete wing of an aircraft was modelled from drawings in CATIA. Linear FEM analysis was performed in CATIA and results were compared with the actual experimental results obtained through static testing of aircraft wing. Same model was then analysed using ANSYS software and results were compared with CATIA analysis.

Key-Words: -Aircraft, Wing, CAD, CATIA, FEA, ANSYS,

1 Introduction

Computer-Aided Design (CAD) has wide applicability in the design and development process of an aircraft[1,3]. CAD is the use of computer software and systems to design and create 2-D and 3-D virtual models, the benefits of which are increasing rapidly, ranging from shape visualization to its analysis, machining, layout designing and considerable cost reduction. Availability of CAD models reduce the experimentation considerably and aid in quick changes to design with initial estimates hence reducing the design and development cycle[2,4]. Creating drawings, preparing reports of assembly and part drawings, preparing bill of materials, etc. become much easier and faster with the use of CAD systems [5,7].

2 Modeling of Wing

Complete wing of an aircraft was modeled in CATIA software consisting of spars, ribs, wing fuselage attachments, aileron and flap attachment, skins panels, stringers and wing skin (lower and upper). Control surfaces were not modeled.A total of 83 parts were modeled.

For convenience a reference drawing was created marking the location of all the ribs and spars as shown in Figure-1. All the members were then modeled with the help of this reference drawing for accurate positioning of parts in the assembly phase. Top to bottom approach was chosen for modeling of wing. The aerodynamic airfoil was chosen as the outer skin and all members of the wing were protruded with reference to this surface[6,8]. Actual aircraft outer contour and drawings were used for modeling of the parts. Sample modeling of wing attachment spar and rib is shown in Figure-2.

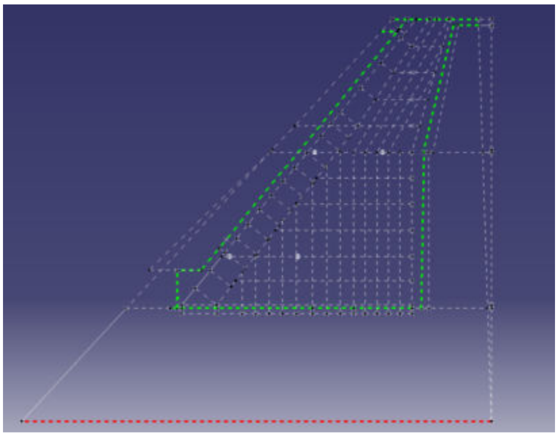


Figure-1: Reference Drawing of Wing in CATIA

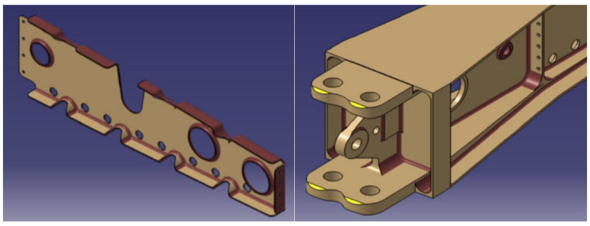


Figure-2 A Rib and Main Spar

All the individually modeled parts were assembled as shown in Figure-3 and 4.

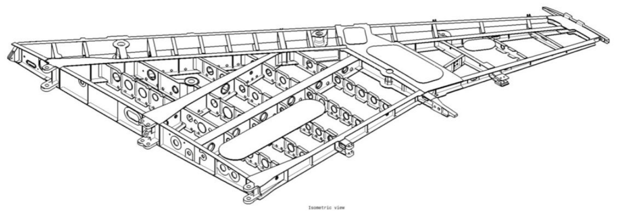


Figure-3. Assembly of Wing Parts

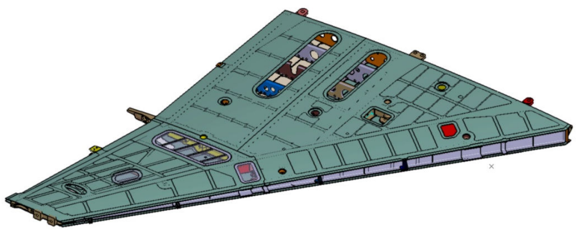


Figure-4. Complete Skinned Model of the Wing

3 Fem Analysis

Finite Element Analysis is a computer simulation technique used in engineeringanalysis that uses a numerical technique called the Finite Element Method[6,8]. FEM is a numerical method which gives an approximate solution to a problem. It is done by dividing the problem domain into several elements. FEM analysis tells us whether a design will fail under given conditions or not before the actual physical product is ever really made.

FEM analysis was done by using FEM solver of CATIA® and ANSYS® software for individual components. The results were compared with the experimental results. Initially single parts were compared using different load conditions. After the results had been verified, the parts were assembled in the assembly workbench and detail analysis was performed on the complete wing assembly.

4 Element Selection

The choice of element type was either to use Hexagon or Tetrahedron element type. Since the analysis was carried out in the elastic range of the materials; therefore, only two element types were used:-

a) Linear tetrahedron (4-node)
b) Parabolic tetrahedron (10-node)

5 Mesh Optimization

Mesh optimization of the rib was done in order to reduce the mesh size and to optimize the results[9,10]. Mesh optimization graphs for both types of elements are shown in Figure-5. It can be seen that considerably less number of elements are required for optimization in case of parabolic tetrahedron elements as compared to the linear tetrahedron elements.

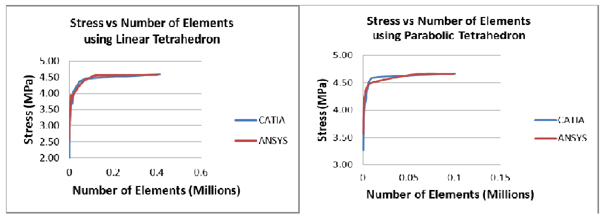


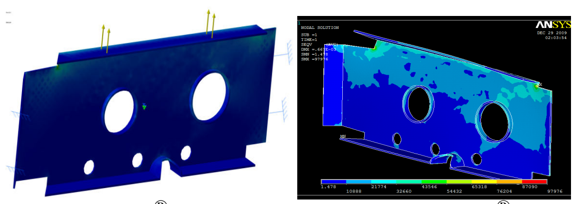
Figure-5: Mesh Optimization ofa Rib

6 Comparison of Analysis Solutions

In order to verify and check the results of CATIA® analysis different analysis cases were run. This was necessary so that it was verified that the correct boundary conditions were specified. For this purpose, analysis of single part was done initially. The analysis was done in CATIA® andANSYS® software. The results from both softwares were compared with theoretical results as well. An analysed case is presented below.

6.1 Analysis on a Rib

A modelled rib was analysed both in ANSYS® and CATIA®. Comparison of results was then carried out. Cleaning was not required after importing of model to ANSYS® as actual model was already simplified by removing all the small curves and fillets. Model was imported as *.model(V4) in ANSYS®. An arbitrary but same load was applied. Resulting stress contours are compared in Figure 6, and stress concentration is shown in Figure 7.



CATIA® ANSYS®

Figure-6: Comparison of stress contours results

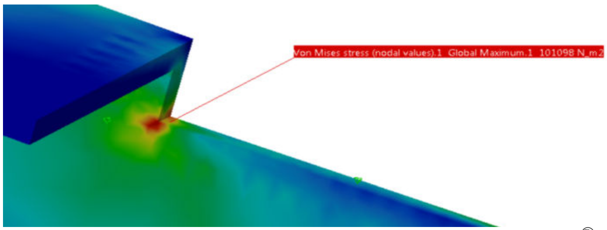


Figure-7: Enlarged image of max stress (CATIA®)

The maximum stress areas and the values came out to be very close and comparable. Two different element type, linear tetrahedron and parabolic tetrahedron were also compared, both in CATIA® and ANSYS®. It was found that parabolic tetrahedron was more accurate and converged much more quickly.

6.2 Analysis on a Complete Wing

After the CATIA®FEM solver had been verified for its accuracy and linear analysis, FEM analysis of complete wing assembly was carried out. The wing was imported to the analysis workbench of CATIA® and different element sizes were given to each component depending on its size. Due to higher computational power required for parabolic element type critical members were identified and given parabolic element type whereas rest were given linear element types[6,8]. Loads from actual static test load data were applied on the wing assembly. Optimized mesh of the de-skinned wing is shown in Figure 8 and its mesh optimization curve is shown in Figure 9.

Figure-8: Optimized meshed model of the wing

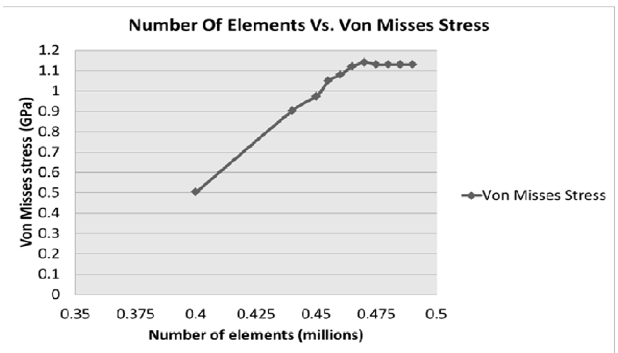


Figure-9: Mesh optimization curve

Maximum stress areas were identified and the mesh was refined locally till stress values converged. Meshes were joined together using "General Analysis Connection" in CATIA® connections menu and rigid connection property was given to them. The wing was constrained in all DOF from the wing fuselage attachment.

Analysis was also carried out for a 2D model of the same wing in ANSYS® software. Similar boundary conditions were applied to the model. Results were comparable in both software with an error of 3.88%. Stress contour is shown in Figure 10. Maximum stress came out to be on the wing fuselage attachment of the main spar for the selected load case shown in Figure 11.

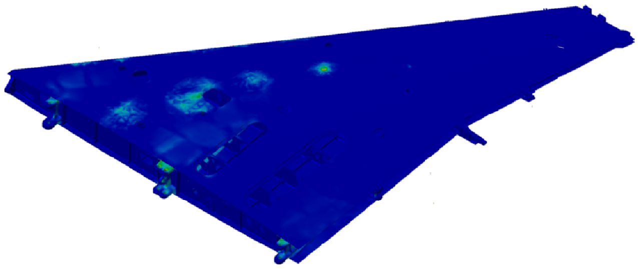


Figure-10: Stress contour of the wing

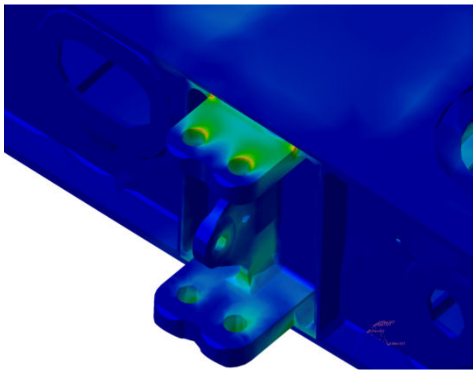


Figure-11: Enlarged image of maximum stress concentration area

4 Conclusion

CAD modeling of the complete wing of a fighter aircraft was done in CATIA® along with FEM analysis. The model was also analyzed in ANSYS® software for comparison of results. The results from both the software were also compared with actual experimental static testing results with an error of 5%. With same computing time the results were reasonably close in both the software. For complex geometries, a small deviation of results was observed due to the simplifications done for the ANSYS model. Although, CATIA® is not a pure FEM software; however, the results were

ISBN: 978-1-61804-193-7

comparable with ANSYS® software for linear analysis. The distinct advantages of modeling in CATIA® were found to be accuracy of model, parametric modeling, Linear FEM analysis, ease of machining, and generation of a layout model. CAD model in CATIA® allowed rapid changing of design during the design stage and its preliminary linear analysis, saving both time and reducing costs considerably.

References:

[1] Brunetti G., Golob B., Computer Aided Design, 2000, 32(14), 877-887

[2] Andreas Krumbein, eN transition prediction for 3D wing configurations using database methods, and a local, linear stability code, Elsevier; Aerospace Science and Technology 12 (2008) 592–598 (J)

[3] HuiHua, et al An experimental investigation on the aerodynamic performances of flexible membrane wings in flapping flight Elsevier; Aerospace Science and Technology 14 (2010) 575–586 (J)

[4] X. William Xu, Tony Liu: A web-enabled PDM system in a collaborative design environment, Robotics and Computer Integrated Manufacturing, 19 (2003) 315–328

[5] Riaz Ahmad, Yuqing Fan, Lu Hu, Analying Innovation, knowledge management and PLM connections, Proceedings of 6th intconf CAID & CD, 2005,Delft, Netherlands; P642-647.

[6] Huang, G.Q.; Lee, S.W.; Mak, K.L., Int. Journal of Advanced Manufacturing Technology, 2001,18(8), 605-613

[7] FarhadAmeri, and DebaDutta, Product Lifecycle Management: Closing the Knowledge Loops, Computer-Aided Design and Applications, Vol. 2, No. 5, 2005, pp 577-590.

[8] Michael Chun and Yung Niu, Airframe stress analysis and sizing, Book, 2nd Edition, 1997, HongKongconmilit press

[9] Vsevolod I. Feodosiev, Advanced stress and stablility analysis, Feb, 2005, Springer-Verlag (Book)

[10] Hakim S. Sultan Aljibori, Finite Element Analysis of Sheet Metal Forming Process, European Journal of Scientific Research, Vol.33 No.1 (2009), pp.57-69