Generative design case study of a CNC machined nose landing gear for an unmanned aerial vehicle

I Zaimis¹, E Giannakis¹ and G Savaidis¹

¹ School of Mechanical Engineering, Aristotle University of Thessaloniki, Thessaloniki GR54124, Greece
E-mail: ioanzaim@meng.auth.gr, efstratg@meng.auth.gr, gsavaidis@meng.auth.gr

Abstract.

Generative design is a design method inspired by nature’s evolution, pushing the limits of the industry to new design solutions. Using minimum input requirements, the resulting designs can be structurally adequate, weight-optimised and almost production-ready. The main purpose of this paper is to present the different approaches of the standard engineering design and the generative design methods. This is achieved through the case study of a nose landing gear for a prototype tactical unmanned aerial vehicle, intended for low volume production. The first part of the paper outlines the conceptual design of the nose landing gear. Subsequently, during the preliminary phase, the stress distribution along the different parts is calculated, based on the classic Strength of Materials theory and therefore a design solution is produced. In the second part, a generative design study is carried out with a commercially available tool, based on the conceptual parameters previously chosen. Both concepts are studied with industry-standard finite element analysis tools in order to validate, from a theoretical standpoint, the strength requirements according to the STANAG 4671 regulation. The final structure will be manufactured with 3-axis CNC machining, while providing about 36% weight reduction without compromising the structural functionality.

1. Introduction

Nose landing gear used in tricycle landing gear configurations, is one of the most critical aircraft parts and is subject to high loads during different mission profiles. The landing gear is one of the last parts to be designed due to its high dependency on the rest of the aircraft parameters, therefore it is important to reach the appropriate design maintaining low weight and adequate structural properties.

In this work, computer aided generative design using the commercially available package Fusion 360 by Autodesk is used to examine the implementation of this method in the design procedure. Using artificial intelligence, many different design concepts can be investigated based on different manufacturing procedures, materials and starting shapes in order to provide the most feasible solution from an early design stage as opposed to other topology optimisation methods [1,2,3]. These options are too many to investigate analytically, but can be easily examined through Fusion 360 [4]. In the meantime, based on a target objective function (minimization of mass, maximization of stiffness) and linear stress calculations, the desirable results are acquired.

The idea that motivates this study is to compare directly the generative design results of a specific manufacturing procedure and material with a standard engineering approach that
uses simple methods according to the Strength of Materials. This is accomplished through the case study of a nose landing gear of a prototype tactical unmanned aerial vehicle designed by Aristotle University of Thessaloniki, shown in Figure 1 and Figure 2. Specifically, the main focus is on the design of the two upper arms and the lower arm. As shown in Figure 3, both aforementioned approaches start with the same concept and preliminary design parameters. During the preliminary phase, the standard approach requires a loop between the analytical and the finite element analysis (FEA), in order to design, improve and validate the arms. The generative approach is more straightforward than the standard one, as it produces organic-shaped, optimised solutions. Both approaches, share the same basic concept, landing gear configuration, location and nose landing gear concept.

2. Conceptual and preliminary design phase
This section outlines the various conceptual characteristics shared by both methods. The unmanned aerial vehicle’s main landing gear is located near the center of gravity while the
secondary landing gear is located at the nose module, as shown in Figure 4. The location is largely affected by various driving stability constraints. The share of the load received by the nose landing gear is 9.5-15% (of the total static weight of 255 kg), securing smooth functioning, ease of direction control and avoidance of porpoising [5,6].

The nose landing gear is a fully articulated arm mechanism, providing high wheel stroke in the limited available space for the landing gear system. In that manner, a custom oleo-pneumatic shock absorber is chosen to maximize the efficiency and maintain a lower stroke.

![Figure 4. Landing gear location.](image)

Throughout the preliminary design of the mechanism, the different load case scenarios are determined based on the relevant airworthiness regulation STANAG 4671 [7]. These loads are shown in Table 1 and they are applied at the wheel. During the level landing scenario the load has the maximum magnitude, while during the side and forward loading scenarios the structure is subject to different load directions. These ground loads are based on two load factors, representing the external forces divided by the weight. The limit load factor refers to the most adverse landing conditions, corresponding to a vertical sink speed of 3m/s, while the ultimate load factor (1.5 times the limit load factor) is used for the structural integrity verification of the landing gear through experiments [8]. These factors can be selected through the energy equation (1).

$$E_{kinetic} + E_{dynamic} = E_{absorber} + E_{tyre}$$

![Figure 5. Geometric Model.](image)

| Axis | Level landing (N) | Brake Roll (N) | Aft (N) | Forward (N) | Side (N) |
|------|------------------|----------------|--------|-------------|---------|
| X    | 995              | -              | 650    | -324        | -       |
| Y    | 3436             | 1912           | 812    | 812         | 812     |
| Z    | -                | -              | -      | -           | -       |

Table 1. Ground loads calculated with the limit load factor.

Figure 5 shows a geometric model created in order to define all these parameters, considering the structural constraints of the mechanism and the requirements of each component. Lastly, other important components are the wheel and the steering mechanism.

The three arms are designed with the material Al7075-T6. The stress design limit that has been set for the limit load factor is the endurance limit of the material, 175 MPa and for the ultimate load factor the yield limit, 503 MPa. The manufacturing method chosen for both approaches is the three-axis CNC milling.
3. Standard design approach

The standard design approach is based solely on the design engineer’s concepts, analyzed using Strength of Materials theory and refined using computer programs. Following the definition of the various parameters, each part is studied individually, based on the worst-case scenarios. According to the regulation, during the level landing condition, the maximum load starting from 25% up to 100% compression of the stroke, is applied.

Firstly, the free body diagram created is shown in Figure 6. The lower arm is designed as a rectangular beam with variations across its length in order to work properly with the rest of the components. In order to calculate an approximation of the dimensions, this study considers four different cross sections along the component, as shown in Figure 7. Based on these sections a major stress distribution is calculated, aiming not to exceed the endurance limit.

Each upper arm is calculated as a combination of three beams, simulating a frame. In order to mount the various components (pin, shock absorber, shaft) with ease and maintain a simple manufacturing process, the upper arm is designed using a simple rectangular cross section 13mm wide.

As the design is quite simple, a buckling analytical calculation under the ultimate loading can be easily executed, considering for each of the lower and upper arms a constant cross section. The minimum safety factor is 8 and is located at the lower arm.

Finally, a FEA is carried out in order to refine and validate the resulting parts. The analysis is static and linear using four CAD models, the two upper arms, the lower arm and the main shaft. The shock absorber is simulated as rigid trush element. The components are easily mapped using brick elements, due to their simple surfaces and geometry. In order to connect the parts, various connector elements and techniques are used such as HINGE elements, trush elements and beam elements with high stiffness, imitating RBE2 connectors. The load is applied at the location of the wheel and the boundary conditions simulate two bearing mountings on the shaft.

The calculation results are plotted in Figures 8 and 9. The maximum stress is calculated in accordance to the TRESACA criterion and is located at the lower arm during the fully compressed level landing scenario with an amplitude of 168 MPa. The maximum vertical deflection at the
wheel is 4.1 mm. Under the ultimate loading the maximum stress is 253 MPa. The final weight for each upper arm is 0.56 kg and for the lower arm 1 kg.

**Figure 8.** Level landing finite element results.  
**Figure 9.** Side load scenario finite element results.

4. Generative Design
The generative design, using Fusion 360, is carried out for each part individually. Only a small portion of important data is required to start the process. As shown in Figure 10 and Figure 11, the design space should be firstly defined. The green shapes determine the regions where material should be preserved.

The algorithm connects these regions together with material paths in an efficient way, based on linear FEA calculations and the level set method. The red shapes act as obstacles, where material paths should not interfere with these regions. Obstacles are defined so that the

**Figure 10.** Generative design model in Fusion 360 for the lower arm.

**Figure 11.** Generative design model in Fusion 360 for the upper arm.
different components will have the desirable clearance and specific mounting regions. Then, the different load scenarios are defined. The parts should be placed in parallel with the directions of manufacturing. Therefore, the forces acquired by a free body diagram are transposed into the manufacturing tool’s coordinate system. Based on the forces obtained from the free body diagram, and the constraints defined for each individual part, the level landing scenario for 25% and 100% wheel vertical stroke is considered. The material is set to be Al7075-T6, with a stress fatigue limit of 175 MPa, and the objective function is set to minimize mass. Lastly, using CNC three-axis machining, the lower arm will be machined using 4 directions, +Y, +Z, -Y, -Z, while the upper arm using two, +Z, -Z.

Regarding the lower arm results, as shown in Figure 12, the algorithm connected the preserve regions in a way that material is distributed furthest from the lengthwise plane, in order to receive the load more efficiently, resisting bending and stiffening the component. In the ”A-A” cross section, a C-type cross section is designed, in a more efficient way than the standard approach, which would have used rectangular cross sections. In the ”B-B” cross section, the same principles apply, with the lower part being closer to the neutral line, acting as the main structural component, and the upper parts provide support against buckling. The upper arm, as shown in Figure 13, shares the same principles, while in the region around the ”A-A” cross section the width of the part varies along the load paths between the preserved regions.

At the present time, the Fusion 360 package has not yet supported symmetry axis definition, and buckling calculation. Therefore, the results used to verify the structure through FEA are slightly modified using the T-spline environment provided in Fusion 360 design module, in order to produce a symmetric component, and fix numerical errors.

![Figure 12. Generative design results of the lower arm.](image)

To verify the results, the parts are exported as CAD files and analysed with the finite element method. As the produced surfaces are quite complex, tetrahedral second order elements are produced by an automatic mesh generator. The connections between the components, the constraints and the loads are identical to the standard engineering approach. The results are shown in Figure 14 and Figure 15. The maximum stress under the limit load is 174 MPa, while under the ultimate load, 264 MPa. The vertical displacement at the wheel is 3.4mm.

For the buckling calculation under the ultimate loading of the new components, a linear finite element buckling analysis is conducted, taking into consideration the first 6 buckling modes. The
results shown in Figure 16 are scaled up 10 times for better visibility. The two upper arms are studied together using beam elements to simulate pins and their safety factor is 1.56, while the lower arm is studied on its own with a safety factor of 2.77.

Each upper arm weighs 0.375 kg, and the lower arm weighs 0.6 kg. This is a reduction of 25% and 46%, respectively, while the overall reduction is 36%.

The results should provide the design engineer with a starting design point. Redesigning the
Figure 16. Generative design buckling modes results. Three upper arm’s buckling modes (a-c) and three lower arm’s buckling modes (d-f).

CAD model will provide simpler surfaces to manufacture and analyse using the finite element method.

5. Conclusions
This paper outlines a design path for the structural design of components using CAD generative design. This is compared with a simple standard engineering approach. Both methods are validated by means of the finite element method. The standard method provides results based on the experience of the individual. It requires more design time, while the manufacturing time is reduced because of the simpler geometry. The generative design method reduces the design time significantly, while also investigates many different concepts and provides lighter designs (in this case by 36%). The results are based on linear FEA, providing efficient material distribution, only limited by the manufacturing capabilities. As generative design evolves, and in view of isogeometric analysis and 3D-printing establishing themselves in the industry, the present study shows the potential of this technology and the benefit of the design procedure.

References
[1] Munk D, Auld D, Steven G and Vio G 2019 On the benefits of applying topology optimization to structural design of aircraft components Struct. Multidisc. Optim. 60 1245
[2] Bagassi S, Lucchi F, De Crescenzo F and Persiani F 2016 Generative design: Advanced design optimization processes for aeronautical applications Proc. Int. Cong. of the Aeronautical Sciences (Daejon)
[3] Kallioras N and Lagaros N 2020 DzAIN: Deep learning based generative design J. in Procedia Manufacturing 44 591
[4] Buonomici F, Carfagni M, Furfari R, Volpe Y and Governi L 2020 Generative design: an explorative study Comp. - Aid. Des. and Appl. 18 144
[5] Pazmany L 1986 Landing Gear Design for Light Aircraft vol 1 (California: Pazmany Aircraft Corporation)
[6] Sarigiannidis K Design, Finite Element Analysis and Optimization of an Unmanned Aerial Vehicle’s (UAV) main landing gear based on a composite leaf spring 2019 (Thessaloniki: Aristotle University of Thessaloniki, Laboratory of Machine Element and Machine Design)
[7] UAV Systems airworthiness requirements (USAR) for north atlantic treaty organization (NATO) military UAV systems, 2007 (NATO Standardization Agency)
[8] Jakubowski R and Tywonik A 2016 An energy absorption dynamic test of landing gear for 1400 kg general aviation aircraft J. of KONES Powertr. and Transp. 23 159