Efficient Shape Optimisation of an Aircraft Landing Gear Door Locking Mechanism by Coupling Abaqus to GENESIS

Mark Arnold and Martin Gambling

Penso Consulting Ltd
GRM Consulting Ltd

Abstract: The objective of this work was to minimize the mass of a mechanism for locking aircraft landing gear doors whilst ensuring stresses did not exceed the material allowable stress. A typical metallic locking mechanism comprises a frame attached to the aircraft structure and supporting a number of linkages through pivot pins. These linkages transfer translational motion of a hydraulic actuator into rotation of a hook, which engages with the door. The structure was modelled with Abaqus/Standard as a non-linear static analysis involving contact, but with no material or geometric non-linearity. Existing commercial optimisation codes are available to perform Topology and Shape optimisation with Abaqus, however these require a non-linear Abaqus simulation to be performed for each design iteration. To achieve an optimized design in a shorter duration, the non-linear Abaqus model was coupled with an equivalent linear VR&D GENESIS analysis model, representing only the frame, using an interface developed by GRM Consulting Ltd. Shape optimisation studies were performed using this coupled approach to derive the optimum geometry for the frame. Whilst optimizing within GENESIS the Abaqus interface periodically executed a non-linear Abaqus simulation with updated geometry to maintain correlation between the two solvers. In conclusion, this coupled optimisation allowed a detailed design to be achieved in a significantly reduced timescale due the efficient application of optimisation technology, using only the minimum number of time-consuming, non-linear iterations.

Keywords: Aircraft, Coupled Analysis, Design Optimisation, Finite Element Analysis, Minimum-Weight Structures, Optimisation, Non-Linearity, Process Automation, Stress Analysis.

1. Introduction

Due to the shortening of aerospace product development cycle times, aggressive cost and mass targets and the demand for increased fuel efficiency, there is increasing pressure upon aerospace engineers to create more efficient designs for aircraft structures and get these to market quicker. Because of these demands, optimisation techniques are becoming commonplace in aerospace engineering since they offer engineers the opportunity to develop minimum mass solutions in a reduced timescale.

The objective of this work, which was performed for a UK aerospace company, was to minimize the mass of a typical aircraft landing gear door locking mechanism whilst ensuring it achieved certain structural targets. Penso would normally have optimized such a static structural problem using the Vanderplaats Research & Development, Inc (VR&D) GENESIS finite element analysis and design optimisation software, which can perform size, topometry, shape, topography and also topology optimisation. However, as this problem required the simulation of contact nonlinearity and GENESIS was restricted to linear static analysis, Abaqus/Standard was selected as the most appropriate finite element analysis software. Abaqus had limited design optimisation capabilities, so the ideal solution would be to link the non-linear analysis capabilities of Abaqus with the optimisation capabilities of Genesis. Fortunately, GRM Consulting Ltd, who distributes GENESIS in the UK, had recently developed software for performing a coupled optimisation between the LS-DYNA non-linear dynamic finite element analysis software and GENESIS and was keen to extend its capabilities to include Abaqus.

2. Coupled Optimisation Approach

To allow efficient design optimisation of the aircraft landing gear component and other such problems, a pioneering approach has been developed coupling the advanced simulation capabilities of Abaqus with VR&D GENESIS. GENESIS is a world leading integrated analysis and optimisation tool supporting topology, size and shape, topography and topometry optimisation methods for linear analysis problems.

The objective of the approach is to allow Abaqus analysis problems considering loading regimes including contact, pre-loads, interference fits and material non-linearity to be efficiently optimised. Currently, methods such as Design of Experiments and Response Surface Approximations are available for problems such as shape optimisation, however, these methods are limited in the number of variables that can be considered and require many non-linear analysis to obtain the required design sensitivity information. For example, a 6 variable DoE problem would require approximately 40-50 Abaqus simulations.

A method has therefore been developed to couple the non-linear analysis capabilities of Abaqus to the design optimisation toolset within GENESIS. The process works by automatically interpreting the non-linear loading in Abaqus into an approximate linear analysis problem within GENESIS. This analysis can then be efficiently optimised using gradient based methods before an updated Abaqus model is automatically generated and analysed to assess the non-linear performance. Due to the approximation to a linear analysis load case some deviation will occur between Abaqus and GENESIS and therefore an iterative loop is defined between the solvers until convergence occurs. This process is demonstrated in Figure 1.

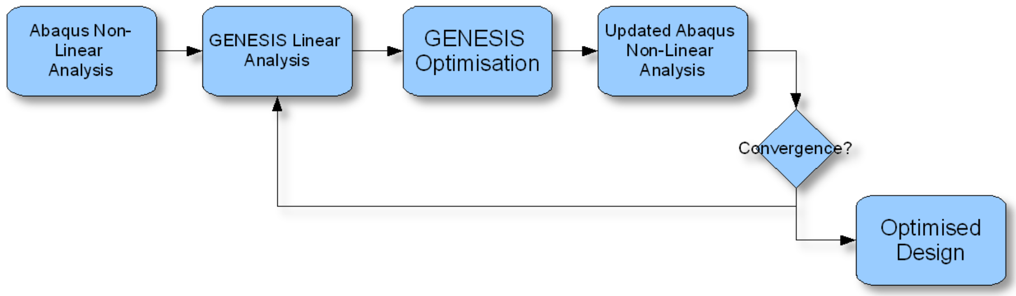

Figure 1. Coupled Optimisation Workflow.

By coupling to the solver within GENESIS sensitivity calculations are made via gradients in the linear solver and, therefore, there is little or no limitation on the number of design variables that can be considered. Whilst in this paper only shape optimisation is considered, methods such as Topology optimisation where tens or hundreds of thousands of variables are present can be considered. Typically, using this new coupling method Abaqus analysis problems can be optimised in anywhere from 2 to 10 non-linear analysis iterations.

By coupling the advanced analysis capabilities of Abaqus to GENESIS it is possible to efficiently consider non-linear design optimisation problems of a size not currently possible using methods such as Design of Experiments and response surface approximations. Some of the key non-linear loading conditions that can be efficiently considered are:

- Contact
- Interference fits
- Pre-loads
- Plastic deformations

Figure 2 shows an example of how the process can be applied to perform topology optimisation on a component loaded via contact and considering interference fits.

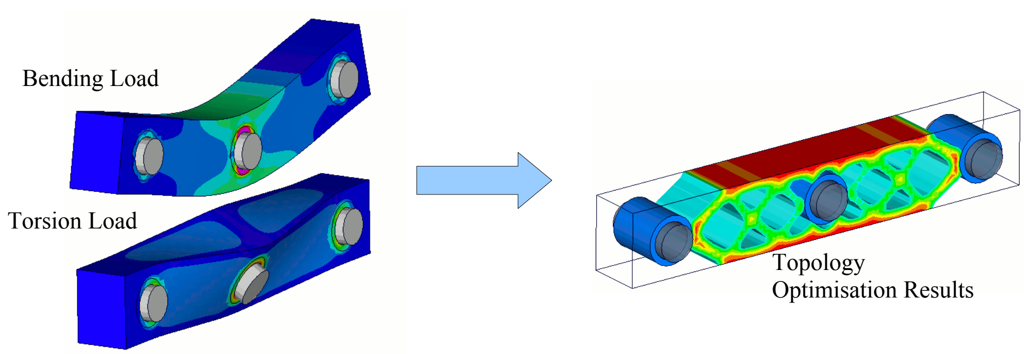

Figure 3. Topology Optimisation Considering Contact and Interference Fits.

The coupling optimisation approach does, however, have some conditions which cannot currently be considered. Once such case is the controlling of plastic strains or non-linear stresses to a defined limit during the Abaqus non-linear loading. Developed to support such constraints is ongoing, however, it is not currently available.

3. Application to aircraft landing gear door locking mechanism

3.1 Description of Uplock Mechanism

To validate this coupled optimisation approach, it was applied to the problem of reducing the mass of an aircraft landing gear door locking mechanism. A typical mechanism comprises a frame attached to the aircraft structure by a number of bolts, as shown in Figure 3. The frame supports a number of linkages through pivot pins, which rotate within bushes pressed into the frame. These linkages transfer translational motion of a hydraulic actuator, mounted to one side of the frame, into rotation of a hook. During extension of this actuator, the hook rotates into a locked position and engages with a roller mounted to the landing gear door as it closes. During retraction of this actuator, the hook rotates into an unlocked position and disengages with the roller, allowing it to open. An alternate actuator, acting through an additional 'alternate' linkage, was mounted to the opposite side of the frame to unlock the landing gear doors in case the primary actuator failed.

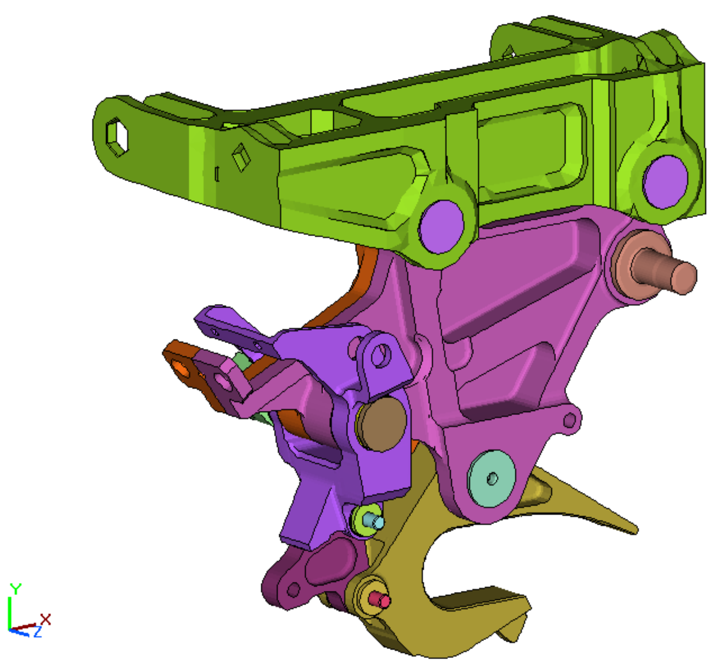

Figure 3. Uplock Mechanism Model.

3.2 Abaqus Input Model Creation

The uplock mechanism frame, linkages and hook were each machined from billets of steel having a number of different grades dependent upon the required yield stress and ultimate stress allowables. The bushes and pins were also manufactured from steel. Static loads applied to the uplock mechanism were classified as either a hook load applied by the roller to the hook, an actuator extension or retraction fatigue loads applied to the linkage, or a combination of these loads. The hook load was further classified as either a limit, ultimate or fatigue loading condition, whilst the actuator load was further classified as either a fatigue or system loading condition. These loads represented events such as tyre burst, flight manoeuvres and a frozen roller.

An Abaqus/Standard model of the uplock mechanism structure was created using ANSA, a pre-processor suitable for use with a variety of finite element analysis software developed by BETA CAE Systems S.A. The uplock mechanism model comprised a mesh of first-order solid (continuum) hexahedral, wedge and tetrahedral elements, with modified second-order tetrahedral elements used to model a section of the aluminium aircraft structure supporting the frame. Distributing coupling constraints were defined on the hook surface to distribute the concentrated hook load over a narrow band of elements, whilst kinematic coupling constraints were used to transfer the single point constraint reaction forces into the aircraft structure and distribute the load generated by the actuator (not modeled) to its attachment pins.

The steel and aluminium materials were modeled using linear elastic and isotropic properties, since previous analysis had shown that the stresses were below yield. As well as not modelling any material non-linearity, geometric non-linearity was also not considered since previous analysis had shown the displacements and rotations to be small. Since the bushes, pins and links were all modeled as unconnected components due to their complex interaction, contact between adjacent components was modeled with pairs of contact surfaces defined on the exterior (free) faces of the solid elements. Therefore, a non-linear static analysis would be required. Tied contacts were also used to constrain some of the nuts and washers to the pins. The finite-sliding, node-to-surface contact formulation was used throughout and friction was not included on the contact surface interaction properties. A number of spring elements were modeled to eliminate rigid body motion of some components, although contact stabilization (**`*CONTACT CONTROLS, STABILIZE`**) was also used in Abaqus to address this issue when contact was not fully established.

Since most of the bushes were press fitted into the frame and links due to being an interference fit, preloads would exist in the frame, links and bushes. Since it was considered important to represent this pre-stress, the first load step of the analysis was to simulate the interference fit prior to applying any external loading. This was achieved in Abaqus by defining an automatic shrink fit to removes these initial 'overclosures' between the bushes and frame/link contact pairs over the step (**`*CONTACT INTERFERENCE, SHRINK`**). The second step of the analysis then involved applying the hook and/or actuator load as a concentrated nodal force. The boundary conditions for both steps consisted of zero displacement constraints at a number of nodes on the aircraft structure in translational and/or rotational DOF.

3.3 Abaqus Baseline Results

The baseline model was analyses using Abaqus/Standard Version 6.7-5 and was subsequently postprocessed using μETA post-processor, also developed by BETA CAE Systems S.A. The nodal averaged, corner von mises stress in the frame for the ultimate hook load are shown in Figure 4. The peak stress occurred at the upper lugs, where the frame was mounted to the aircraft structure, with a slightly lower stress at the lower lug, about which the hook pivots. Although the peak stress exceeded the material ultimate allowable, this was as a result of the aircraft structure bushes being physically connected at the nodes, instead of using a contact, and so this stress was ignored. The mass of the baseline frame was 0.0609 lbf sec^2/in (a mass unit consistent with inch, sec, lbf).

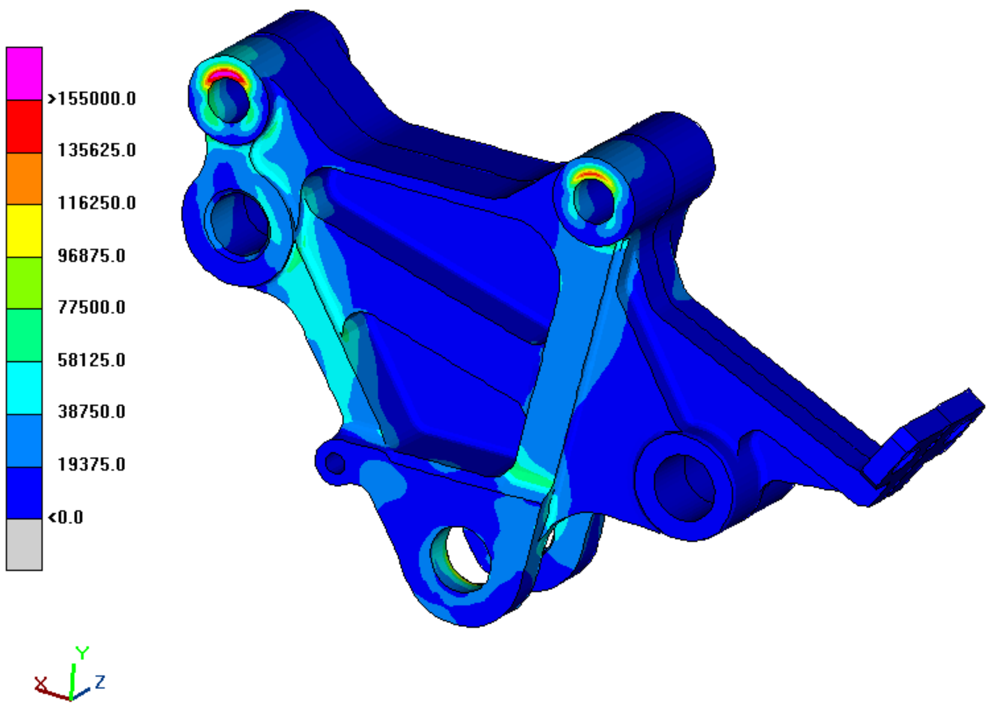

Figure 4. Von Mises Stress in Frame before Optimisation.

3.4 Coupled Optimisation Setup

Since the intension was only to optimize the frame, the first stage was to create an input model of the frame in GENESIS format, which is very similar to that of MSC NASTRAN. This was easily achieved by importing the Abaqus model into ANSA and then exporting just the frame nodes and elements in NASTRAN format. It was important that the numbering of the node and element ids was identical in the Abaqus and GENESIS models so the coupled optimisation software could correlate the results from both sets of analysis.

The second stage was to import the NASTRAN input file of the frame into Design Studio, the pre- and post-processor for GENESIS. Since the frame would not have any single point constraints defined, inertia relief constraints had to be defined to restrain the model and eliminate rigid body motion. It was decided to optimize the thickness of the vertical and the inclined members connecting the lower lug to the two upper lugs, since the stress in these members was found to be low from the baseline analysis. Since the frame model comprised of solid elements, this would require a shape optimisation involving node perturbation to adjust the size of these members in the analysis model.

Shape optimisation domains, three-dimensional hexahedral volumes, were defined following the feature lines of the frame and each containing the solid elements bounded by the volume (see Figure 6). Perturbation vectors were then applied to the corner points (nodes) of these domains to adjust the thickness of the two frame members within user-defined limits through the use of two design variables. The lower bound for the thickness was defined to be greater than the minimum member size for machining.

The objective function for the optimisation was to minimise the mass of the frame . A constraint was defined on the nodal stresses with an upper limit of 155000 psi, which was the material ultimate stress allowable.

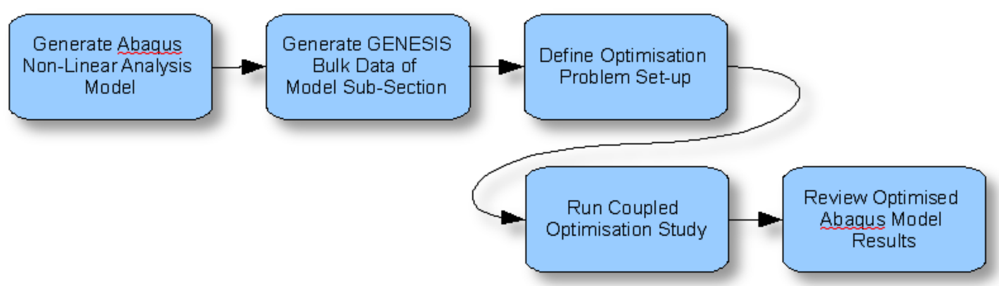

Figure 5. Coupled Optimisation Workflow.

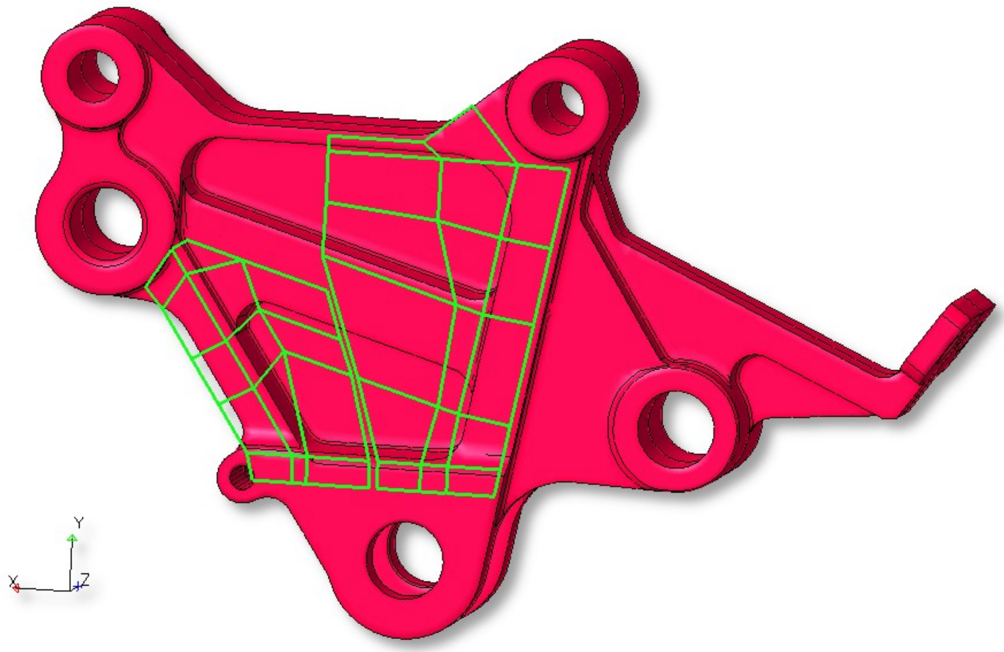

Figure 6. Frame Shape Optimisation Domains.

3.5 Coupled Optimisation Results

The nodal averaged, corner von mises stress in the frame for the ultimate hook load after optimisation are shown in Figure 7. The peak stress again occurred at the upper lugs, where the frame was mounted to the aircraft structure. As the objective of the optimisation was to minimise the component mass, whilst ensuring stresses remained within the limits, the stress in the two members connecting the lower lug to the upper lugs was noted to increase. They did, however, remain within the defined stress limit of 155000 psi. The mass of the optimised frame was 0.0582 lbf sec^2/in, showing a notable reduction on the baseline design. It is also important to note that the shape optimisation design variables, by definition, ensured that the manufacturing dimensional constraints were satisfied.

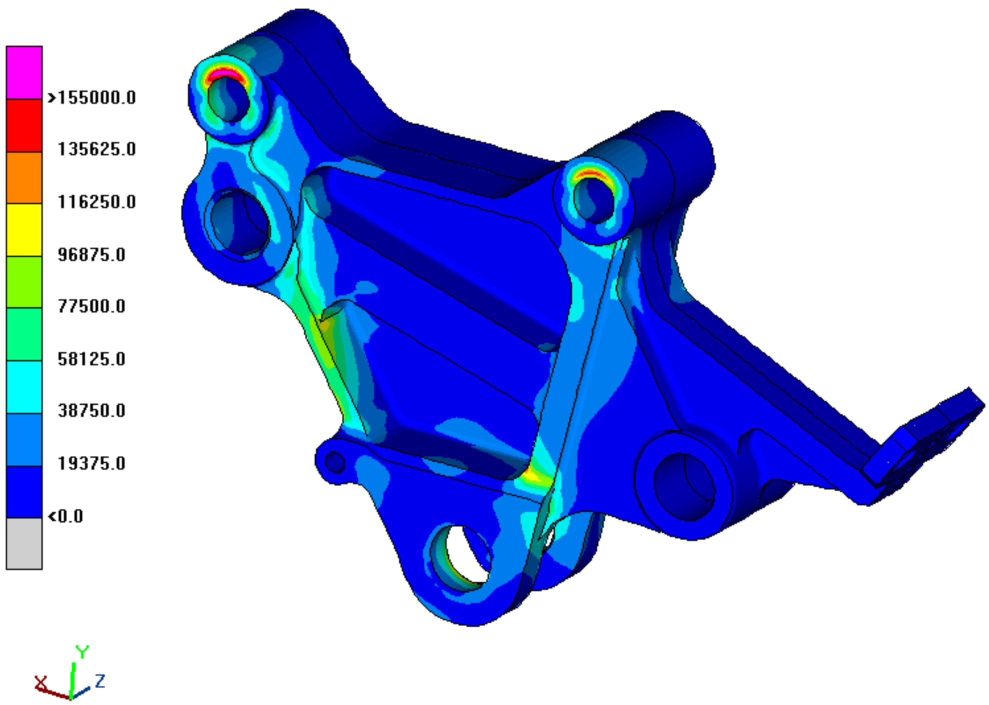

Figure 7. Von mises stress in frame after optimisation.

4. Conclusions

The coupled optimisation was found to allow a mass optimized design to be achieved in minimum time and within defined design constraints due to a greatly reduced number of costly non-linear design iterations.

The coupling of GENESIS' efficient linear optimisation to the advanced analysis capabilities of Abaqus has proved to be an extremely efficient method for design mass reduction. The coupling of methods such as Topology optimisation to Abaqus capabilities such as contact, pre-load and plasticity brings about a new advance in what can be achieved through design optimisation tools, saving both development time and cost through lighter solutions, brought more quickly to market.

In conclusion, a power solution has been developed, which allows engineers to utilise the most appropriate analysis software for loading assessment (eg Abaqus, LS-DYNA, MARC, PAM) whilst still being able to make use of the well proven optimisation capabilities of GENESIS.

5. References

1. Abaqus Analysis User's Manual, Version 6.7-5, Simula, 2008

2. VR&D GENESIS Design Reference Manual, Vanderplaats R&D Inc, Version 10.0, 2008

3. GRM Abaqus Solver Coupling Interface, GRM Consulting Ltd, Version 1.3, 2009