

Aircraft Landing Gear Simulation using Abaqus/Explicit

Acknowledgement

Dassault Systèmes SIMULIA would like to acknowledge Fokker Landing Gear for providing the geometry and model parameters used in this article.

Summary

A mechanical system, such as aircraft landing gear, can have a large number of parts that interact in a complex nonlinear fashion. The challenge of simulating such a system lies not only in capturing the correct physical behavior but in using efficient analysis techniques. Different levels of modeling abstraction may be appropriate for different stages of the design process. Initial sizing and kinematics can be studied with a partially rigid representation, while final designs are more often analyzed with fully meshed flexible geometry.

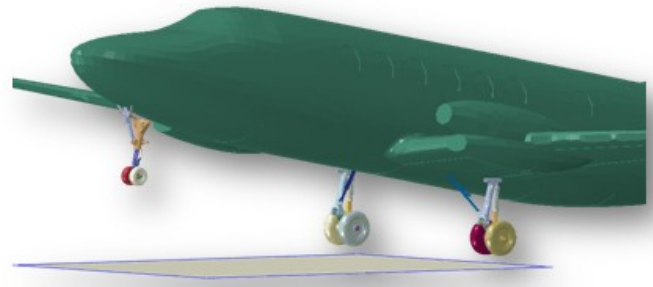
The use of Abaqus/CAE and Abaqus/Explicit to develop a process for predicting landing gear performance is outlined in this Technology Brief. By combining the automation and customization capabilities of Abaqus/CAE, the robustness of Abaqus/Explicit, and Unified Finite Element Analysis (UFEA) concepts, we show how the Abaqus product suite can be used to efficiently analyze mechanical systems.

Introduction

The landing gear of an aircraft is a complex system that has many components. Traditionally, landing gear numerical simulations have been carried out with Multi-body Dynamics (MBD) software.

While Abaqus has all of the necessary capabilities to perform MBD simulations, it additionally has the ability to extend a rigid body kinematic model by exchanging a rigid representation of an individual component with its elastic counterpart. This allows you to selectively or completely include flexible components for stress calculation and nonlinear behavior. Consistent with the philosophy of UFEA, a single model can be used for several different analyses.

One challenge in the development of landing gear is the initial design and sizing of the system. While some components and designs can be reused, the overall layout and sizing is unique for each application. The customization capabilities of Abaqus/CAE allow you to build plug-in applications that can automate some parts of design process.



Key Abaqus Features and Benefits

- Ability to customize Abaqus/CAE with plug-in applications that can accelerate and automate the design process
- Connector elements for modeling mechanical joints, mechanisms, and point-to-point representations of structural components
- Capability to combine rigid and flexible parts in the same model, allowing for varying levels of abstraction during the design process

Modeling and Analysis Approach

A generic aircraft, with two main landing gears (MLG) and a single nose landing gear (NLG), is shown in Figure 1. Two factors are considered in simulating the landing: velocity and pitch angle. When contacting the ground, the loads from the aircraft are transferred into the landing gear through the attachment points with the airframe. The primary quantities of interest are the forces in the landing gear strut components.

Two approaches to modeling the landing gear are taken. Both use connector elements to simulate the mechanical joints where the individual parts of the gear are assembled. Figure 2 illustrates some of the most frequently used connector types in Abaqus.

The distinction in the two approaches lies in the way the individual parts comprising the nose and main gear (struts, actuators, springs, dampers, etc.) are represented.

In the first approach, rigid bodies are used to represent the components of the gear. In conjunction with the connectors at the mechanical joints, this allows us to use a



Fig. 1: Aircraft landing gear system

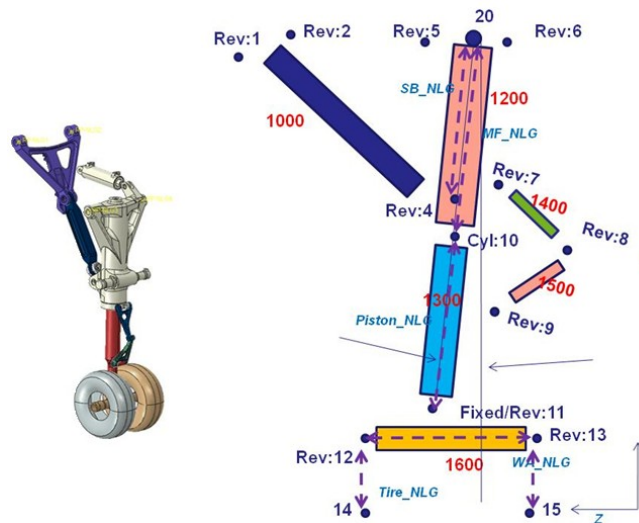


Fig. 3: Schematic diagram of the parameterized landing gear

Assembled	Basic translational	Basic rotational
BEAM	LINK	ALIGN
WELD	JOIN	REVOLUTE
HINGE	SLOT	UNIVERSAL
UJOINT	SLIDE-PLANE	CARDAN
CVJOINT	CARTESIAN	EULER
TRANSLATOR	RADIAL-THRUST	CONSTANT VELOCITY
CYLINDRICAL	AXIAL	ROTATION
PLANAR		FLEXION-TORSION

Fig. 2: Partial catalog of connectors in Abaqus

to different designs. The struts and joints are given specific element numbers. The schematic representation of the NLG is shown in Figure 3. With this approach, one can very easily model landing gear systems for different types of aircraft and perform design studies with less turn-around time.

To facilitate the construction of the parameterized model, an Abaqus/CAE plug-in has been developed. The plug-in has a user-friendly interface that helps visualize the role of each parameter. A screenshot of the plug-in is shown in Figure 4.

The joints are modeled by JOIN, REVOLUTE, CYLINDRICAL and other types of connectors. The spring and damper is modeled by a CYLINDRICAL-type connector

multi-body modeling approach. Here, the connector elements provide an easy and versatile way to model physical mechanisms with point-to-point geometry while still capturing complex kinematic and kinetic behavior. This approach is particularly appropriate for the early stages of the design process.

In the second approach, a mesh is put on the actual geometry of one of the gear components. The clear advantage to this approach is the ability to compute stress and deformation distributions in the parts. Given the increased computational cost of this approach, is it more appropriate for analyses in the later stages of the design process.

In this work we present a fully parameterized model that allows easy changes to dimensions and other properties. This way, CAD geometry does not need to be modified every time a new set of parameters is used. To achieve this, a “nodes-and-elements centric” modeling approach is adopted. Specifically, each attachment point between the landing gear and the aircraft is given a specific node number and has a coordinate that can change according

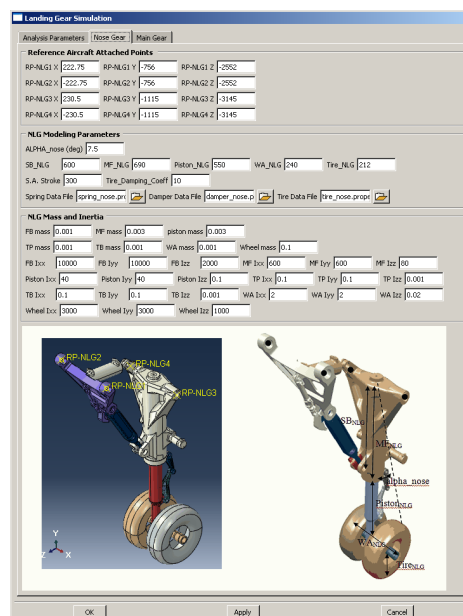


Fig. 4: Abaqus/CAE plug-in for building a parameterized landing gear model

that has nonlinear elasticity and damping properties. Tires are modeled with connectors defined with a “stopping” behavior that is triggered when the tire is in contact with the ground.

Multi-body Dynamic Analysis

In the first part of the analysis sequence, an MBD simulation is performed in Abaqus/Explicit with the gear components modeled as rigid bodies. After performing a convergence study of the results, a time increment size of $2e-5$ is chosen and used for all further simulations.

To demonstrate the present approach, we choose to study the design change that is shown in Figure 5. Specifically, the angle of the main fitting component of the MLG is changed from 4° to 2° and the angle of MLG shock absorber is changed from 9° to 5° . Other variables and simulation conditions are identical. The chosen aircraft landing conditions include a 10 degree pitch angle, a 2.4 m/s downward velocity and a 20 m/s forward velocity.

Figure 6 shows the effects of the design change on NLG and MLG shock absorber total forces. Results with a suffix of “MD” indicates the modified design. We can see that with the modified design, the NLG touches ground at a later time.

Flexible Body Dynamic Analysis

In the second part of the analysis, the main fitting component of the MLG is replaced with a deformable finite element model and the same dynamic analysis is performed. Figure 7 shows the Mises stress contour in the MLG component during landing. Figure 8 compares the MLG shock absorber total force between the rigid and flexible body analyses.

Conclusion

A parameterized modeling approach for analyzing aircraft landing gear in Abaqus/Explicit and Abaqus/CAE is presented. The methodology demonstrates how customization and efficient modeling techniques can be used to accelerate the design of mechanical systems. By taking advan-

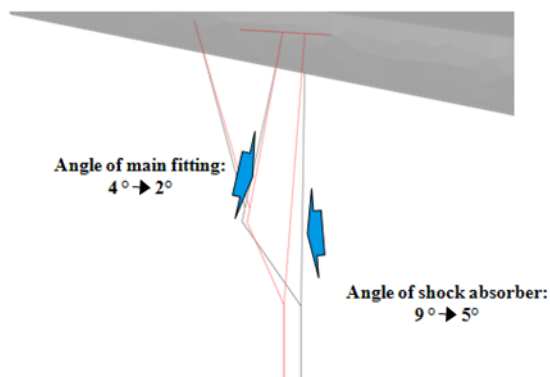


Fig. 5: Main landing gear design change

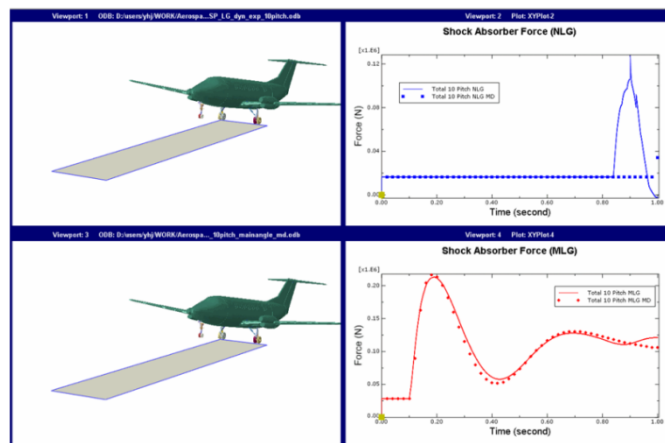


Fig. 6: Shock absorber total forces in NLG and MLG

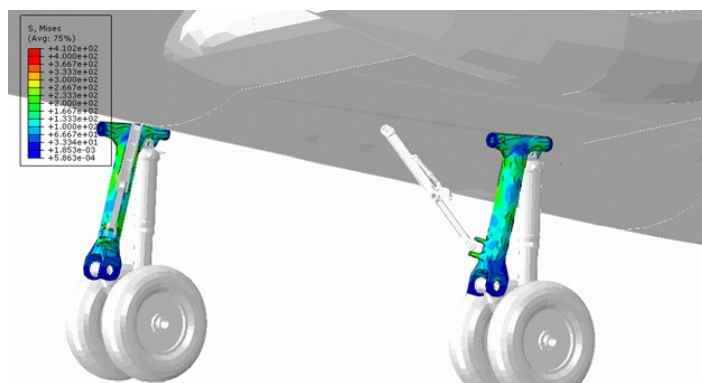


Fig. 7: Mises stress contour in the MLG main fitting component

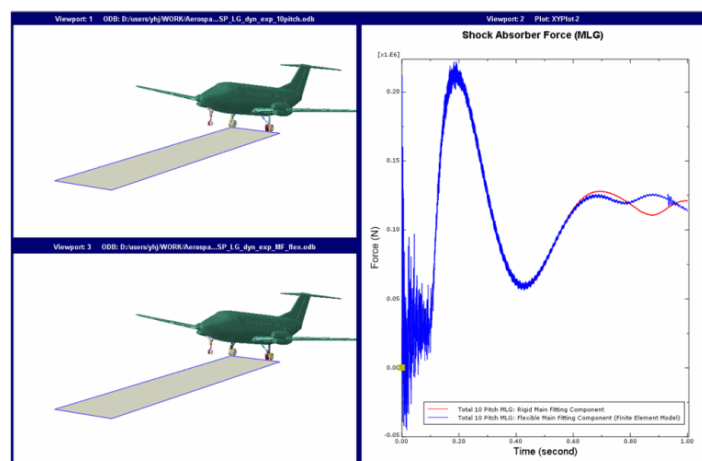


Fig. 8: Comparison of shock absorber force between rigid and flexible body simulations

tage of Abaqus' ability to use a combination of connectors, rigid, and flexible part representations, a single model can be used to quickly size the mechanical system and explore different combinations of physical parameters.

Abaqus References

For additional information on the Abaqus capabilities referred to in this brief, please see the following Abaqus 6.11 documentation references:

- Analysis User's Manual
 - Parametric Input, Section 1.4.1
 - Connector Elements, Section 30

About SIMULIA

SIMULIA is the Dassault Systèmes brand that delivers a scalable portfolio of Realistic Simulation solutions including the Abaqus product suite for Unified Finite Element Analysis, multiphysics solutions for insight into challenging engineering problems, and lifecycle management solutions for managing simulation data, processes, and intellectual property. By building on established technology, respected quality, and superior customer service, SIMULIA makes realistic simulation an integral business practice that improves product performance, reduces physical prototypes, and drives innovation. Headquartered in Providence, RI, USA, with R&D centers in Providence and in Suresnes, France, SIMULIA provides sales, services, and support through a global network of over 30 regional offices and distributors. For more information, visit www.simulia.com

The 3DS logo, SIMULIA, Abaqus and the Abaqus logo are trademarks or registered trademarks of Dassault Systèmes or its subsidiaries, which include Abaqus, Inc. Other company, product and service names may be trademarks or service marks of others.

Copyright Dassault Systèmes, 2007

