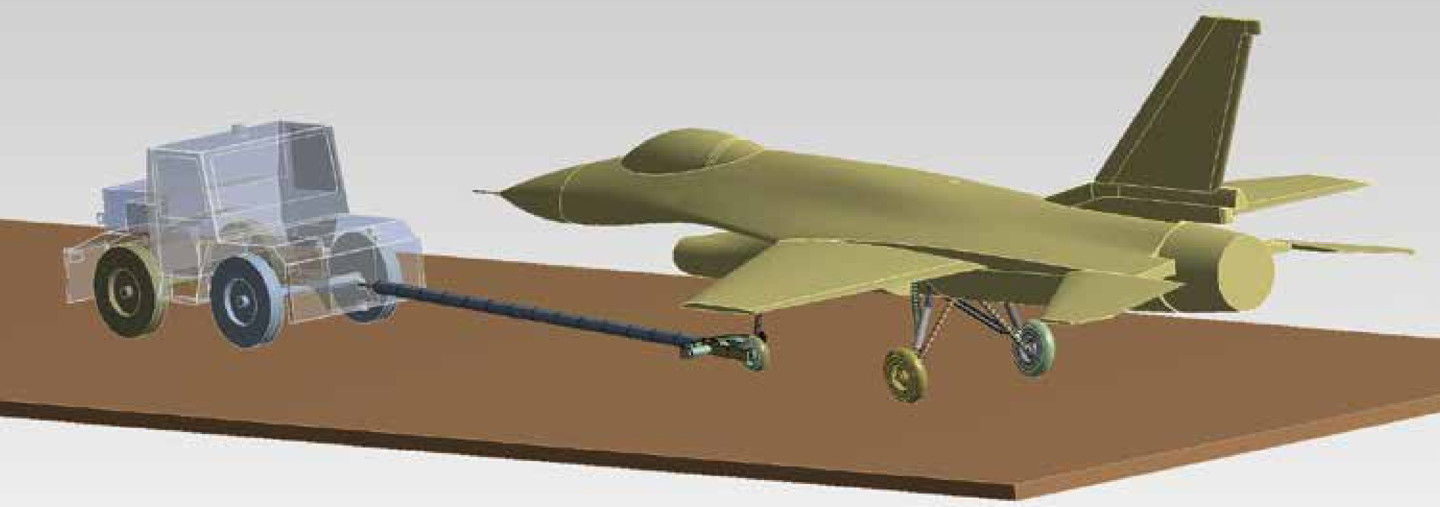


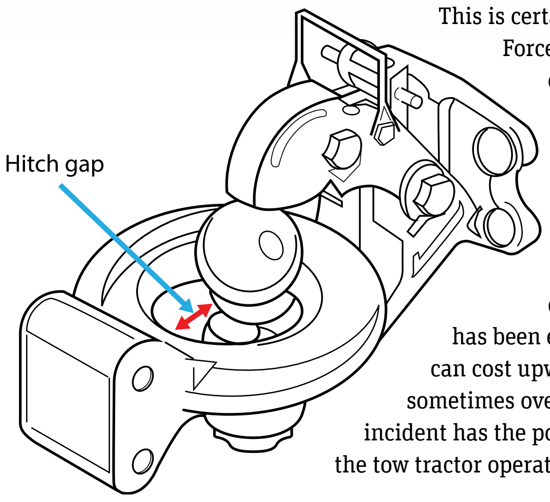
Hitting the Brakes

United States Air Force jets were being damaged when the tow tractor that transports them around bases came to a sudden stop. An Air Force engineering team used ANSYS Mechanical to determine the cause of the problem and devise a simple solution to this multimillion-dollar problem.

By **Andrew Clark** and **Jared Butterfield**, Lead Structural Analysis Engineers, United States Air Force, Hill AFB, USA

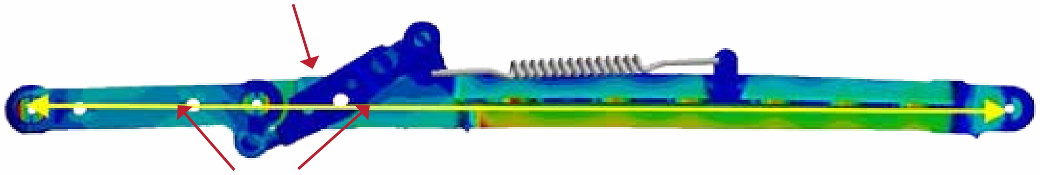
Because affordability is one of the key mantras of the U.S. Department of Defense, and engineering for sustainability initiatives (to optimize operational availability of assets while controlling costs) is growing in importance, engineering simulation is playing an increasingly significant role.

This is certainly the case at the United States Air Force (USAF). Before a fighter jet can take off to perform its mission, it must be towed from the maintenance shed to the hangar, from the hangar to the taxiway, etc. USAF lightweight jets experienced mechanical damage after impact loads from a tow bar connection exceeded design limits during a sudden stop by the tow tractor. It has been estimated that a single failure of this type can cost upwards of a million dollars. The aircraft sometimes overhangs the tow tractor, so this type of incident has the potential to cause death or serious injury to the tow tractor operator, not to mention damage to and loss of



“Fifteen separate transient *dynamic analyses* were completed to simulate the *various combinations* of factors.”

operational capability for the aircraft. Engineers were puzzled because the drag-brace assembly — the landing component that originally failed in these accidents — should have been designed to withstand known tow-bar loads. Physical testing of the aircraft was of limited use in determining the cause of the problem because an actual aircraft could not be risked in a test. The USAF team solved the problem by simulating a wide range of braking incidents to determine the conditions under which the drag-brace assembly could fail so they can be avoided in the future.



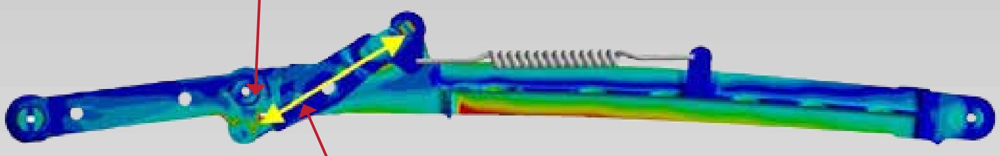
^ Initially, the upper drag brace bends, resulting in column instability.

SIMULATION HELPS DETERMINE ROOT CAUSE

The USAF team first performed finite element analysis (FEA) with ANSYS Mechanical on the drag-brace assembly to determine whether or not it was strong enough to withstand the towing limit loads in the design specification. Engineers created a model of the drag-brace assembly and performed a static structural analysis that showed that the assembly is even stronger than the design specification. The actual drag-brace assembly was placed in a test fixture and loaded in accordance with the FEA simulation. The test results agreed with the structural simulation and demonstrated that the assembly indeed exceeded the design specification. Simulation and testing further defined the sequence of events that occurs during failure. First the upper drag brace bends, resulting in column instability. Next, the primary load path changes to a secondary and weaker load path involving the smaller downlock link assembly. This secondary path load overcomes the downlock link lug, causing the drag brace assembly to fail catastrophically.

Next, engineers performed a multibody simulation using the ANSYS Mechanical Rigid Body Dynamics add-on module for ANSYS Workbench to quantify the loads imparted to the drag-brace assembly when the tow tractor driver hit the brakes. They modeled the towing assembly using CAD software, then imported the geometry into ANSYS Workbench and created a finite-element model using line, shell and solid elements. Material properties including modulus of elasticity, Poisson’s ratio and lumped mass or density was incorporated into the model to account for stiffness and inertial effects. Spring stiffness and damping properties were defined for the nose and main landing gear struts. These properties were applied as user-defined joints to the struts as a function of position and velocity. The tow bar was attached to the tow vehicle with a translational joint using constraint equations that simulated various sizes of hitch gap — the distance between the tow vehicle pintle hook and the tow bar ring. The tow bar connects to the drag brace assembly in the landing gear to tow the aircraft; the hitch gap is the play or slack in this connection. The stiffness of the tires of the fighter jet and tow tractor were included in the model using information provided

Multibody Dynamics: Rigid and Flexible Methods
ansys.com/multibody



^ After the drag brace bends, the primary load path shifts to the down lock link assembly.

by the tire manufacturers. Engineers used time-history velocity data acquired from physical testing as an input to the simulation to increase the accuracy of the load response. Velocity and braking frictional forces were idealized as linear over time.

PARAMETRIC STUDY

Engineers recognized that variable impact loads could occur with different tow tractors, at different speeds, with various braking forces, under diverse operating conditions, etc. Some or all of these variables could have a major impact on the loading of the drag-brace assembly. They accounted for this uncertainty by parameterizing variables that they suspected might play a major role in the series of accidents, including tow-tractor weight, velocity, acceleration time, stopping time and hitch gap. Fifteen separate transient dynamic analyses were completed to simulate the various combinations of factors defined during the testing phase of the contract. The results from these fifteen simulations were compared against test data to validate the model.

Engineers concluded that the shape of the braking model depends upon the tow operator. This in turn affects the load response and causes significant variation from event to event. In spite of this, they determined that the maximum compressive force that develops from the impact event was highly dependent on the hitch gap. A larger hitch gap generated higher compressive forces. The simulation showed that when the hitch gap exceeds a half inch, the collision between the tow bar and tow vehicle can generate compressive loads in excess of the drag-brace assembly's ultimate load. Further simulation iterations showed that decreasing the hitch gap reduced loads significantly across all analysis and test conditions. Engineers also determined that the weight of the tow truck had a significant effect, with heavier tow trucks generating greater loads on the drag brace assembly.

Controlling this gap was determined to be a simple and effective solution in maintaining towing loads below the allowable limit. The Air Force recommended new procedures that limit the hitch gap and mandate that only tow tractors less than a specified weight could be used to tow smaller jets. These new procedures will improve safety and eliminate damage to the nose landing gear of these expensive aircraft during towing operations.

This application provides a typical example of how the USAF is using engineering simulation to determine the root cause of performance issues so they can be quickly and efficiently resolved to save money and improve operational readiness.

$3.6 Million Saved in Nose Landing Gear Piston Simulation

In another case, replacement of nose landing gear pistons on a Boeing 707 variant was a major expense. USAF engineers used ANSYS Mechanical for structural and fatigue analysis to identify a new thread repair method that extends the life of these parts. The static strength margin of safety was verified through simulation, and the fatigue life was verified through digital fatigue analysis. Savings are estimated at $2.3 million in avoidance of new procurements and $1.3 million in reduction in repair expenses in the first year of implementation alone.