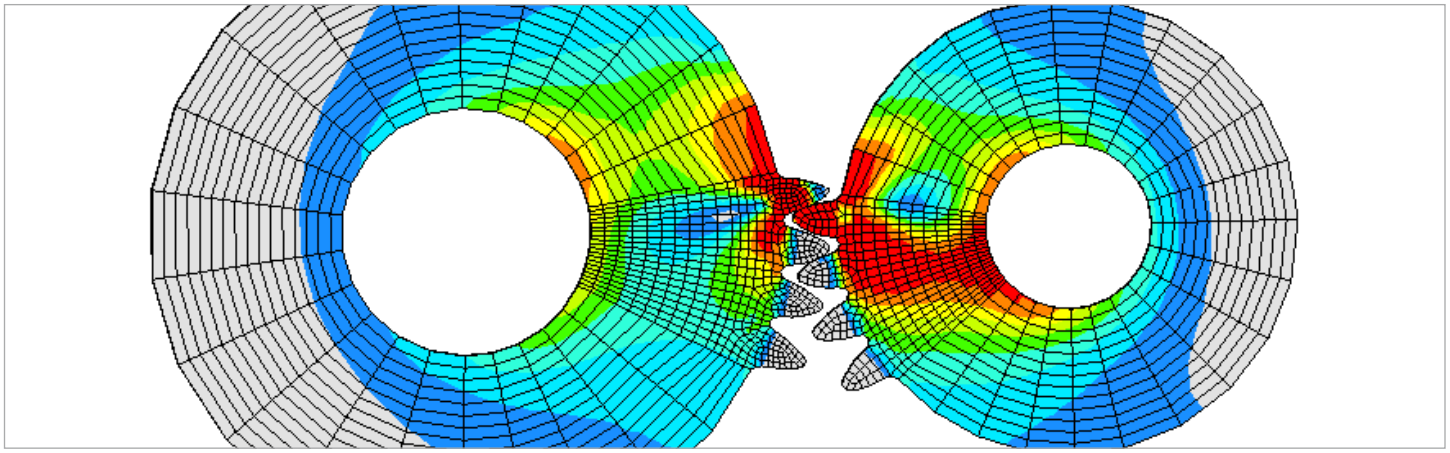
OptiStruct for Structural Analysis: Not Just for Optimizations Anymore



Von Mises stress plot of gears using a sliding contact

There is a trend from computer-aided engineering (CAE) vendors to consolidate tools into one platform.This wave of change is a way for engineering firms to reduce their software budgets, training and web of licenses.

Altair has been a major player in this consolidation space with its HyperWorks platform. But even before the Altair Partner Alliance began, the organization was consolidating CAE functionality within its optimization tool OptiStruct.

“Optimization is always based on underlying structural analysis,” said Uwe Schramm, chief technical officer at Altair. “So, OptiStruct has always been an analysis tool.”

Traditionally, coupling simulation results to a third-party optimization program can be tricky. It involves application program interfaces (APIs), and at times programming, which can slow development down or introduce bugs. Therefore, having the finite element analysis (FEA) tool embedded into the optimization tool can provide a significant advantage.

But that advantage can work both ways. Now an engineer can focus on the optimization first and their validating simulations later. This can create a significant boost in the development cycle and will go a long way toward finding flaws early in a design.

How an Optimization Tool Became a Structural Analysis Tool

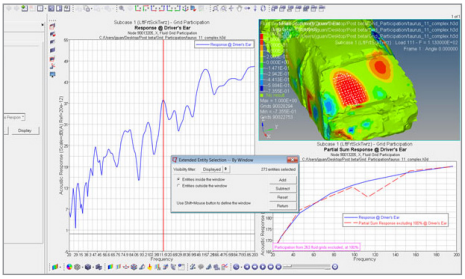
About 20 years ago, what would become the current incarnation of OptiStruct had its start. From this inception, structural analysis was made a priority. NASTRAN is one of the most widely used structural simulation solvers. Therefore, the OptiStruct development team wanted to ensure that its optimization tool was compatible.

As a result, the team decided that it would make sure the inputs for the program were compatible with the format of NASTRAN. The decision paid off as to this day engineers will typically be able to run a NASTRAN solution into the OptiStruct solver.

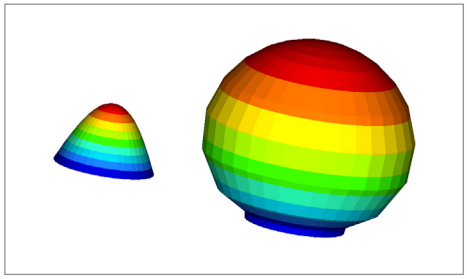
“Over 90 percent of NASTRAN decks run in OptiStruct without modification,” said Schramm “If modifications are required, they do not take long to perform.”

The story shows that the team wanted to not only focus on OptiStruct as an optimization tool; it also wanted to focus on its structural capabilities. Today, the tool has expanded to handle more than just linear static problems. For instance, some of the analysis that engineers can perform in OptiStruct include:

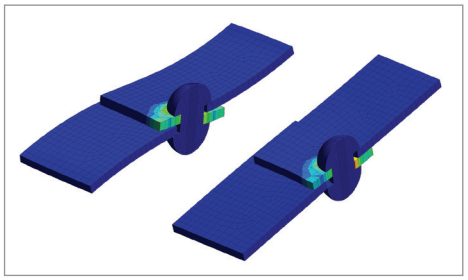
- Linear and nonlinear static and transient
- Heat transfer
- Structural
- Normal modes
- Frequency response
- Complex eigen value analysis (brake squeal)
- Direct and modal transient response analysis
- Noise, vibration, and harshness (NVH) analysis
- Random response
- Response spectrum
- Condensation
- Fatigue analysis



Response study simulated the noise heard in the driver's ear



Geometric nonlinear loads applied to a model using OptiStruct



Small vs. large displacement analysis comparisons can be performed with OptiStruct (Image courtesy of Altair)

Once these analyses are completed, the engineer can then pivot them into an optimization study using a few small steps. First, the engineer adds the design variables they wish to assess. These variables are then governed by constraints and responses that are set by the engineer. Finally, the engineer must set up an objective function that will be used to rate the iterative success of the optimizations.

However, Schramm suggests that engineers instead start off with the optimizations and then use simulations to verify the final designs. He said, "Why bother to check if a design is OK or not, when you can ask the software to give you the design that fits your requirements?"

OptiStruct's NVH Capabilities

Another aspect of OptiStruct that might surprise some engineers is its NVH capabilities. NVH is often used in the automotive industry to simulate the noises that drivers and passengers will experience. Using a modal frequency response, engineers can assess this noise early in a car's development cycle.

The faster these studies are performed, the more optimized the engineers can make their designs. That is why the speed of the NVH solver was a major focus in the release of OptiStruct 14.0.

In fact, the tool has a multilevel substructuring eigenvalue solver (AMSES) that can assess multiple air cavities simultaneously. Some of the software's other NVH capabilities include:

- One-step transfer path analysis
- Modal analysis without eigenvectors
- Error checking and auto-correcting models
- Frequency-dependent bushings
- CDS super elements
- Normal modes that are preloaded by nonlinear analysis

Why Some Engineers Choose OptiStruct for Structural Analysis

Schramm reports that some of Altair's customers have replaced their traditional CAE tools with OptiStruct. Many made this choice to take advantage of both its FEA and optimization capabilities. He also claims that OptiStruct will be quite fast when compared to many pure analysis codes.

Schramm explains that this is because the software is designed to be fast and robust so it can quickly iterate through the optimization process. He said, "With our new Domain Decomposition Method, we can dramatically reduce runtimes while running with over 100 cores at a time. A speed up of over 30x has been achieved when compared to a single core."

While this may be very impressive, many seasoned simulation engineers will note that there are many reasons why organizations will stick with their simulation software. It could be legacy, vendor-specific analysis or functions, training, user interfaces or just plain old preferences that prevent an organization from adopting any new FEA tool, whether that new tool is made by Altair or any of its competitors.

In this case, OptiStruct can still be embedded into an engineering workflow for its optimization capabilities. This is because data inputs for the software are compatible with NASTRAN and the outputs are the familiar PUNCH and OP2 files. As for those not using NASTRAN, they can convert the data into something OptiStruct will understand using a tool within HyperMesh. Additionally, super elements from NASTRAN, ANSYS and ABAQUS can all be imported into the tool.

To learn more about OptiStruct, read an overview of the product here: OptiStruct Overview. Altair has sponsored this post. It has provided no editorial input. All opinions, unless otherwise stated, are mine.

—Shawn Wasserman
Simulation and IoT editor at ENGINEERING.com

Original article on ENGINEERING.com