A Design-Validation-Production Workflow for Aerospace Additive Manufacturing

Megan Lobdell, Brian Croop, and Hubert Lobo; DatapointLabs Technical Center for Materials
Sridhar Ravikoti and Robert Yancey; Altair Engineering Inc.

Summary

The aerospace industry is implementing 3D printing on a selective basis, particularly for one-off and small-lot production, and also to produce parts that cannot be made using conventional manufacturing processes. With 3D printing, there is this incredible possibility to build a solid model, optimize its shape using topology optimization, and then directly print it to make a ready-to-use part. A critical step is to ensure that the part will perform as simulated. This is because simulation includes uncertainty from the assumptions of the model. This uncertainty is often left untested until the prototyping stage. A mid-stage, CAETestbench™ validation is added to the design workflow to measure the simulation accuracy before parts are made, bringing confidence to the engineer's model.

This white paper illustrates the following process workflow:

1. Identify Geometry
   - Create or import geometry as a CAD file
2. Develop material model
   - 3D print test specimens with the actual material and process; measure material properties
3. Validate the simulation
   - 3D print a CAETestBench component (modified Cornell bike crank [1])
   - Test CAETestBench component using Digital Image Correlation (DIC)
   - Simulate CAETestBench component in target solver (Altair OptiStruct®) using HyperWorks® for pre- and post-processing
   - Validate simulation against test and confirm the ability of the FEA to correctly model the 3D printed material
4. Optimize geometry for load case
   - Now use solidThinking Inspire™ to create an optimized design
5. Print the component

Introduction

With the advent of 3D printing and additive manufacturing, manufacturing designs previously thought difficult to produce can now be generated quickly and efficiently and without tooling. In the aerospace industry weight is of great importance to the engineering design. In modern light-weighting strategies that involve geometric optimization, the goal is to generate designs that meet the original performance requirements, but with less material. While in the past, overdesign would be acceptable as a means to prevent part failure, a key requirement of the new scenario is to ensure that the optimized part does not fail. For this strategy to succeed, it is necessary that the simulation be accurate up to failure.

Traditionally, the design process involves much iteration between the designer and the analyst, where the designer submits a design to the analyst, and the analyst completes his or her analysis and sends recommendations back to the designer. The process is then repeated until a valid design meets the analysis criteria. The design is then handed to the manufacturing team, which then may have additional constraints or concerns, and iterations can continue. Additive manufacturing coupled with topology optimization allows the design-and-analysis and manufacturing iterations to be reduced significantly, or even eliminated. To ensure that the part will perform as simulated, a mid-stage validation is conducted on a standardized part before creating the final products. This is because it is often difficult or even impossible to accurately test and replicate the real life boundary conditions of the product with adequate fidelity for quantitative comparison.

Design Constraints

This paper outlines a process that may help to reduce the time required to design, analyze, and produce an aircraft component, while also significantly reducing its weight. The example chosen is a 172 Cessna rear elevator bellcrank[1]. The project goal is to optimize the part for weight based off the maximum allowable load input, while ensuring the part has a Factor of Safety of 2 and a maximum deflection of 3.8 mm.

The existing design, (Figure 1), was first analyzed to generate the baseline engineering performance parameters that would need to be maintained in a new design. The existing bellcrank design was set up for an analysis study by applying a vertical load to the outermost hole, pulling perpendicular to the flat bottom of the bell crank. The vertical load was defined as 2,168.5 N based off the maximum allowable input the pilot can put on the system, as given in FAA Standard S23.397 [2]. The boundary conditions allowed rotation in the center hole, but fixed translation. Additionally, the four smaller holes where the torque tube is mounted were constrained from having horizontal translation and in-plane rotation.

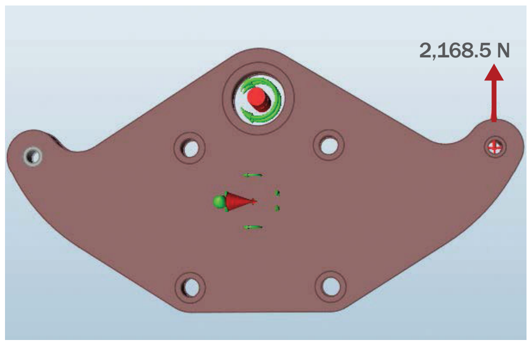
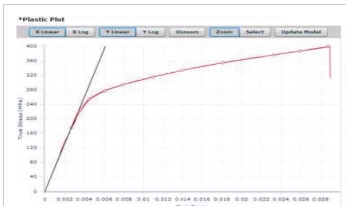


Figure 1. Application of load and constraints on existing bellcrank model.

Develop Material Model with Matereality® Software for Materials

Tensile test specimens were 3D printed using the actual material to be used for the component: AlSi10Mg. Density and stress-strain curves were measured and the resulting data were uploaded to Matereality. Elastic and elastic-plastic piecewise (matx36) material models were created using Matereality's CAE Modeler software. CAE Modeler gives the user graphical control to specify the stress-strain points used in the model and also calculate modulus and ultimate strength for the material model. The material files were stored in the users' Workgroup Material DatabasePro ready for export to HyperMesh through the Matereality-HyperWorks connectivity.



Material	AlSi10Mg	
Young's Modulus	63122	MPa
Yield Stress	213.0041	MPa
Poisson's Ratio	0.331	
Failure Plastic Strain	0.02247295	mm/mm
Density	2.63E-09	tonne/mm^3

Figure 2. Matx36 material model fitting with accompanying linear properties.

Validation of Simulation

Prior to the optimization, a mid-stage validation of the simulation was performed using a carefully conducted test on a 3D printed, standardized Cornell bike crank, which contains features designed to probe the quality of the simulation [3]. Through the use of digital image correlation (DIC), images of the strain field on the face of the Cornell crank were gathered to compare to the simulated strains to evaluate fidelity of the simulation to the test.

[1] Existing bellcrank design model created in house using SOLIDWORKS software, based on measurements taken from a commercially purchased component.

Validation of Elastic Response

The 3D printed Cornell crank model was simulated using the OptiStruct solver and a MAT1 elastic model using Young's modulus, Poisson's ratio, and density values from Figure 2. The simulation was set up in HyperMesh with a solid map mesh at an average mesh size of 0.5 mm creating solid elements. The boundary conditions fixed the surfaces of the rectangular mounting holes. The pin hole was loaded at the center with a force of 650 N.

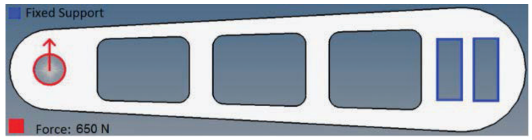


Figure 3. Bike crank boundary conditions.

The elastic simulation used a linear analysis for the OptiStruct solver with the small strain option enabled. As a robust solver, OptiStruct is useful for many linear and nonlinear load cases. Simulation images for the normal elastic strains along the length of the crank were uploaded to PicSci™, an Electronic Lab Notebook by Matereality which allows easy cross-platform documentation and comparison of test data.

Running parallel to the simulation was the physical test to measure the elastic normal strains along the length of the crank. The rectangular holes were mounted in a fixture and the crank was bent using a 650 N force on the pin hole through lifting the pin with a Universal Testing Machine. Strains were measured on the face of the crank using DIC by analyzing the change between stereoscopic images of an applied speckle pattern on the face at each time step during testing. The measurement scales from each strain field image were kept the same to allow for comparison of equivalent contour colors for the range of strain. From the side by side comparison from PicSci (Figure 4) showed the contours matching between images, indicating that the simulation and test produced comparable strains for the applied load.

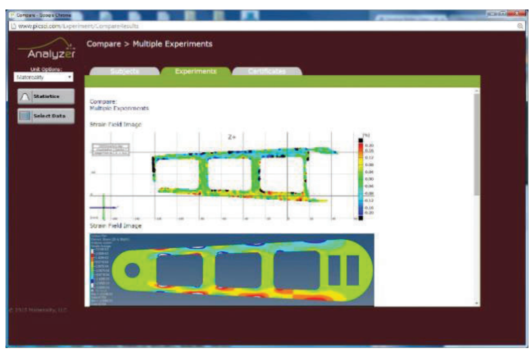


Figure 4. Lab and simulation, comparison of strain fields.

Failure Validation

The crank was mounted and loaded to complete failure on a Universal Testing Machine with force vs. extension measured. In this case, the crank sustained an 1,800 N load before brittle failure occurred in the bottom left corner followed by the upper right corner of that hole. The image of where the crank fractured was uploaded to PicSci for comparison.

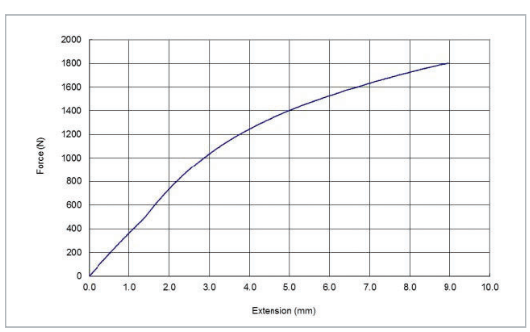


Figure 5. Experimental load curve, force vs. displacement.

The simulation model was also created in HyperMesh, using a refined mesh on the front third by the pin; the refinement is to 0.5 mm, with the rest using a coarser mesh. The mounting holes were fixed, and the load was increased to 1,800 N based on the failure testing. The pin was added to the model, requiring a Type 24 contact setting between the pin and the crank. This case used a tabulated plasticity model to account for the plasticity. Element deletion was enabled based on the failure strains.

The failure simulation used the nonlinear explicit analysis RADIOSS solver. As a robust solver, RADIOSS is useful for many nonlinear load cases handling contact and complex material models. In our case the recorded results were simulation images of the location of predicted failure; these were uploaded to PicSci.

A picture of the broken crank and the simulation image were compared. Not only did they break in the same locations, but the simulation calculated a similar failure load compared to the physical test. This demonstrates the ability of the solver to correctly calculate the strain and failure in the test.

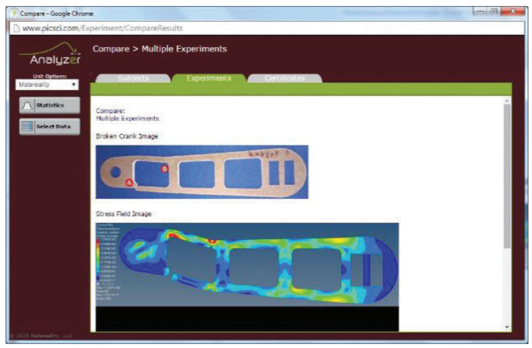


Figure 6. Lab and simulation, comparison of failure of the bike crank.

Geometric Optimization

With a measure of confidence in the simulation now established, an optimization of the elevator bellcrank was commenced using the solidThinking Inspire™ structural optimization software. The resulting conceptual design achieved a mass reduction of 45%, which was significant for any structure and especially for aerospace structures, where minimizing weight is critical.

Using the material properties for the aluminum shown in Figure 2, an analysis was run to generate the baseline results shown in Table 1 for later comparison to the optimized crank.

Max Deflection	0.05 mm
Peak Stress	55.6 MPa

Table 1. Baseline results of existing bellcrank analysis.

Given that the design requirements were that deflection should not exceed 3.8 mm, and that the Factor of Safety was 2 (a peak stress of 107 MPa), it is clear the above design was overdesigned for the requirements, and thus, there was room for optimizing the part for weight reduction.

To begin the optimization study, the design space was defined. The design space essentially is sections of the model Inspire can remove or reconfigure to reach the optimal design. To account for bearing failure, a radius of the non-design space around the bolt holes was determined to be 1.5x the diameter of the bolt holes. The design space is shown in brown, and the non-design space is shown in gray in Figure 7.

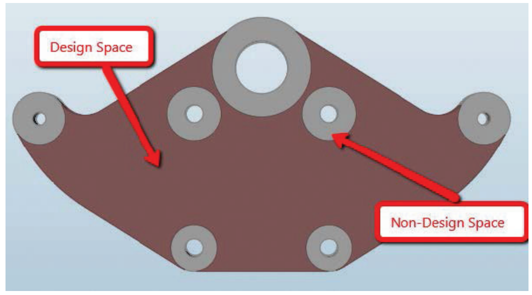


Figure 7. Design optimization model set up showing design and non-design spaces.

The first optimization run set an objective to maximize the stiffness at a target of 30% volume fraction. Volume fraction defines the volume of the original designable space that is available for optimization and 30% is the default value that generally works well in developing the initial concept design. The generated result showed a mass reduction of 54% with a Factor of Safety of 1.6. The resulting concept design is naturally symmetric due to the way the problem was formulated by fixing the center, but one can enforce a symmetry constraint in the design where this is desired.

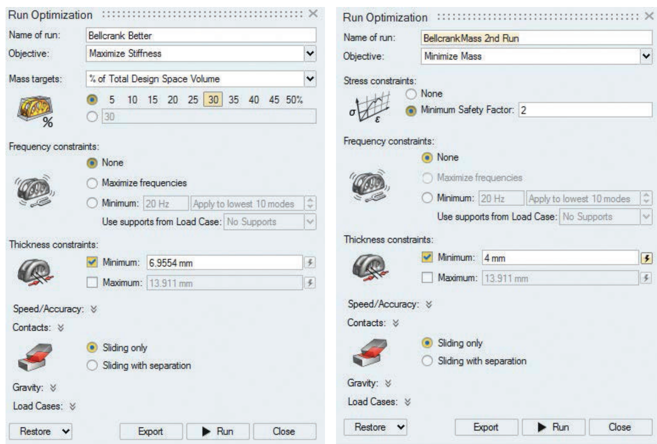


Figure 8. Topology optimization set-up parameters used for initial (left) and final (right) runs.

A second set of iterations were needed to reduce the peak stress and increase the Factor of Safety. After adjusting several of the optimization parameters, a final iteration run generated results satisfactory to the design and analysis conditions. The adjustments included establishing a stress constraint equal to half the yield strength of the material to meet the Factor of Safety requirement, and setting a thickness constraint to establish the minimum thickness that would be allowed in the design. The thickness constraint helped ensure that the stress constraint was met, but is also a parameter that can be used to conform to minimum build parameters in an additive manufacturing process. The problem set-up parameters are shown in Figure 8, and the Factor of Safety results and the derived geometry are shown in Figure 9.

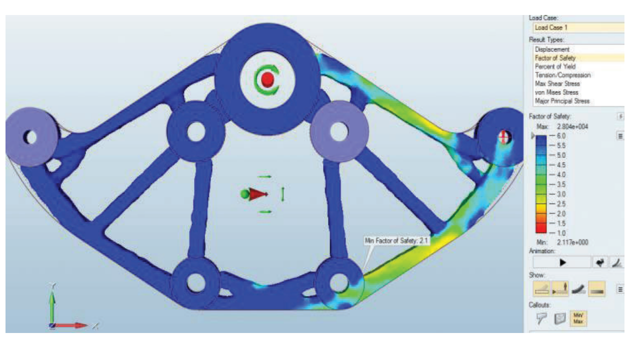


Figure 9. Final topology optimization with a plot of Factor of Safety.

The final optimization iteration generated the following results, as shown in Table 2.

Max Deflection	0.1524 mm
Peak Stress	100.6 MPa
% Mass Reduction	52.6

Table 2. Performance and mass results for final design iteration.

A finalized concept design was generated. The problem remained of how to convert that concept to a feasible geometry for manufacturing and CAD purposes. Inspire has a new feature called PolyNURBs to quickly generate free-form solid geometry that is smooth and continuous from optimization concept results. The PolyNURBs method reduces the time it takes to create geometry to hours or even minutes, instead of day or weeks using older methods. Figure 10 shows the geometry created using PolyNURBs. This is a very smooth geometry that will be free from stress concentrations.

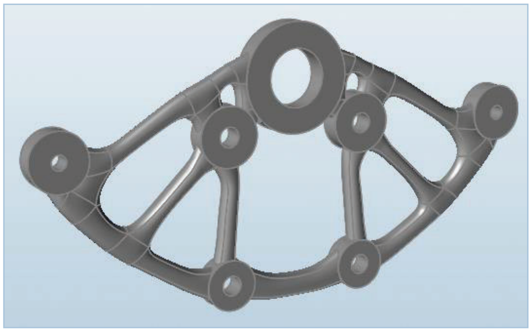


Figure 10. Smooth result interpretation using PolyNURBS technology.

With the system used to build the part, there was concern about "stair-stepping" from the process that could create stress concentrations on the surface. These can be smoothed using sanding, grinding, grit blasting or polishing, but this can take significant time and effort. For this study, it was decided to build and test the part as it comes directly from the machine. In order to do that, the part design was flattened so only vertical walls existed in the structure, which eliminated any stair-stepping from the additive manufacturing build process. The resulting flattened geometry is shown in Figure 11. The corresponding Factor of Safety plot is shown in Figure 12. The analysis of the flattened bellcrank is shown in Figure 13.

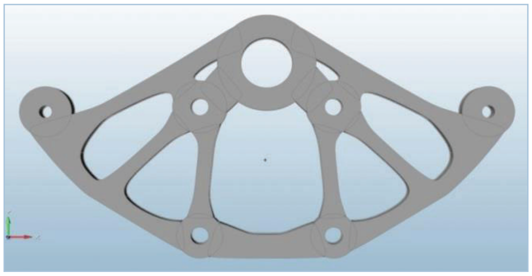


Figure 11. Flattened topology optimization result for build.

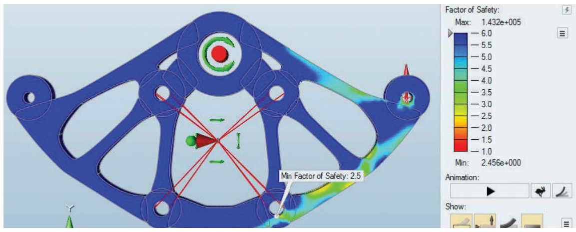


Figure 12. Factor of Safety for flattened geometry.

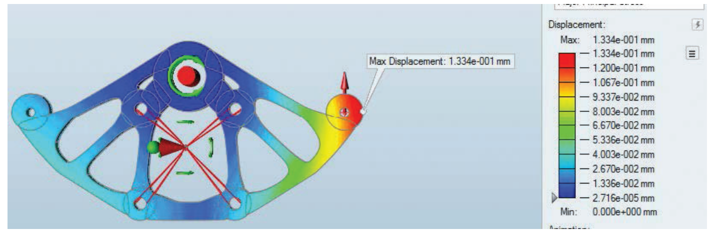


Figure 13. Performance results of flattened geometry showing displacement.

Original Design:	Optimized Design Results:
Max Deflection	
0.0508 mm	0.127 mm
Peak Stress (Von Mises)	
55.6 MPa	86.72 MPa

Table 3. Final performance results of flattened bellcrank.

As shown in Table 3, the deflection and peak stress are higher in the optimized design. This is largely due to the original design having a Factor of Safety that was much higher than typically necessary. In this study, we reduced the Factor of Safety to 2.0, which generally is considered to be very conservative. The final mass results are shown in Table 4. This shows a mass reduction of 45%, which is significant for any structure and especially for aerospace structures where minimizing weight is critical.

Original Mass:		
0.000153	0.138	0.304
tons	kg	lbs
Optimized Mass:		
0.0000841	0.080	0.170
tons	kg	lbs
% Difference		45.00651

Table 4. Final mass results comparing original design with optimized design.

Confirmation

To confirm that the cranks performed as designed, the optimized printed part was deformed in a real-life test, again using DIC to measure surface strains. The resulting strains were compared to those obtained in the optimization simulation.

Three cranks were printed in the XY plane. The printing was performed by Incodema 3D, Inc. (Ithaca, NY) using direct metal laser sintering of EOS Aluminum AlSi10Mg gas atomized powder from EOS GmbH. A 370 W, 100-500 µm variable diameter Yb fiber-laser on an EOS M280 was used to sinter the 30 µm layers. The laser traversed the pattern at a 1,300 mm/s scanning speed with a hatching distance of 0.19 mm. For each layer the scanning path was rotated 66°. After cooling, specimens were cut away from the bed.

The crank was prepared for DIC by painting it with a speckle pattern. It was rigidly held using bolts for the square pattern while the outermost hole was pushed downward via a cylindrical pin attached to the actuator of the load frame as shown in Figure 14. The larger hole is loosely fixed with a bolt to act as a pin. The actuator was moved to apply a force of 2,170 N in the y direction, perpendicular to the crank. The pin in the hole was allowed to rotate freely. The test was performed at a speed of 4 mm/min up to 2,170 N force using an Instron Universal Testing Machine. The force vs. time was recorded as shown in Figure 15. Figure 16 shows a comparison of the DIC measured strains and the predicted strains from the OptiStruct FEA model. There is very good correlation between test and simulation for both peak strains and strain pattern.

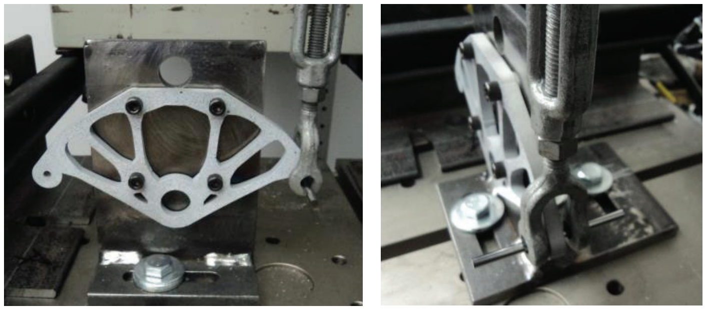


Figure 14. Optimized bellcrank test setup for DIC strain measurement.

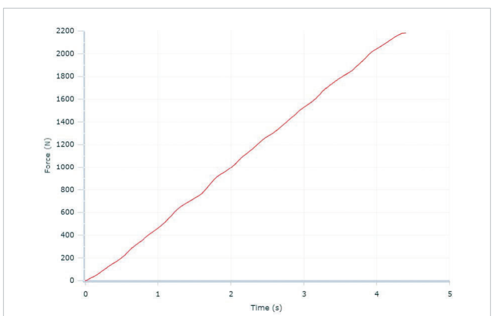


Figure 15. Force vs time curve of the lab experiment.

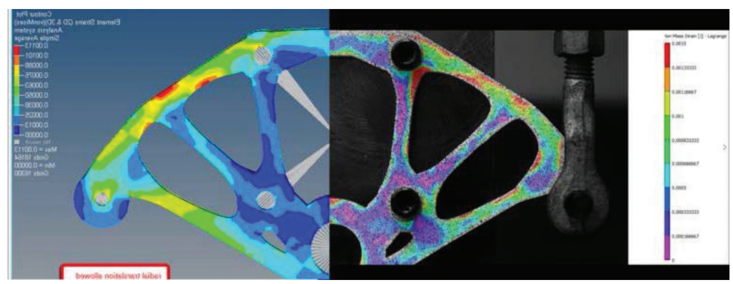


Figure 16. Surface von Mise's strain comparison from simulation and test.

Conclusions

This paper illustrates the workflow of design, analysis, optimization, build, and test for an aerospace component. Incorporating a mid-stage validation into the production workflow is required to confirm the efficacy of the solver and material model prior to use in real-life parts. The produced part will generally have lower weight through an optimized design that will not increase manufacturing costs. Additive manufacturing coupled with topology optimization provides significant opportunities to the aerospace industry because of these benefits. This process can be used as the basis for certification and qualification of additively manufactured aerospace components.

References

[1] Borshoff, J., Roy, D., Croop, B., "The Use of Digital Image Correlation (DIC) and Strain Gauges to Validate Simulation." NAFEMS USA Regional Conference. (2014).

[2] "ELECTRONIC CODE OF FEDERAL REGULATIONS." http://www.ecfr.gov/cgi-bin/text-idx?node=14:1.0.1.3.10#se14.1.23_1397.

[3] Lobdell, M., Croop, B., Ravikoti, S., "Standardized Validation Brings Confidence to 3D Printing." Altair Engineering White Paper (2016)

Acknowledgments

We are grateful to Mr. Kevin Engel and others at Incodema3D for advice on 3D printing and for preparing the test samples.

About

Altair

Altair is focused on the development and broad application of simulation technology to synthesize and optimize designs, processes and decisions for improved business performance. Privately held with more than 2,600 employees, Altair is headquartered in Troy, Michigan, USA and operates more than 45 offices throughout 22 countries. Today, Altair serves more than 5,000 corporate clients across broad industry segments.

Altair's HyperWorks is an on-demand software platform that includes statistical, database, visualization and simulation software to help companies make better business decisions. HyperWorks uses a patented licensing technology allowing customers to transparently share licenses globally across a broad suite of applications.

Altair Partner Alliance

Altair's HyperWorks platform applies a revolutionary subscription-based licensing model in which customers use floating licenses to access a broad suite of Altair-developed, as well as third-party, software applications on demand. The Altair Partner Alliance effectively extends the HyperWorks Platform from more than 20 internally developed solutions to upwards of 60 applications with the addition of new partner applications. Customers can invoke these third-party applications at no incremental cost using their existing HyperWorks licenses. Customers benefit from unmatched flexibility and access, resulting in maximum software utilization, productivity and ROI.

DatapointLabs Technical Center for Materials

DatapointLabs Technical Center for Materials strengthens the materials core of manufacturing enterprises to facilitate their use of new materials, novel manufacturing processes, and simulation-based product development. Materials information is used across the product life cycle for design and product development, prototyping, manufacturing, procurement, and failure analysis. We provide expert material testing, material parameter conversion and model validation services for CAE, allowing companies to populate their databases with high-quality, application-ready data for design and new product development. Matereality® Software for Materials gives companies the means to build databases to store properties, CAE material files, and material information on any material. The built-in suite of web-based software helps engineers visualize and understand material data, create CAE models and manage materials information. We help companies build enduring data collections that accurately represent the materials used in their products.

Altair Engineering, Inc., World Headquarters: 1820 E. Big Beaver Rd., Troy, MI 48083-2031 USA
Phone: +1.248.614.2400 • Fax: +1.248.614.2411 • www.altair.com • info@altair.com