

Iterative Design Evaluation Process in Abaqus for CATIA V5

Summary

During product development, design engineers often have the freedom to modify a number of parameters. However, any design modification requires validation to ensure the satisfaction of requirements for all load cases. With Abaqus for CATIA V5 (AFC), nonlinear finite element technology is made available within the CATIA environment, allowing design engineers to efficiently incorporate accurate stress analysis into the design process.

In this Technology Brief two approaches are described to illustrate the productivity gains possible with AFC. The first uses CATIA's Knowledgeware capability to define parametric studies that assist in making engineering decisions. With this approach, geometry and load parameters can be captured in a design table to further simplify model modifications. The second approach converts proven analysis methods into templates that can be reused to analyze design variations efficiently and with minimal effort. This approach facilitates the replacement of an entire geometric component in the model without having to redefine the analysis attributes. An automotive suspension control arm model is used to demonstrate the two approaches.

Case study: Automotive Suspension Control Arm Assembly

Suspension systems play a major role in vehicle dynamics and driving comfort. The control arm (Figure 1) regulates the motion of the wheel throughout the suspension travel and strongly contributes to the car's performance. The stress in this structure has to be studied carefully for each geometry variation to ensure a satisfactory design.

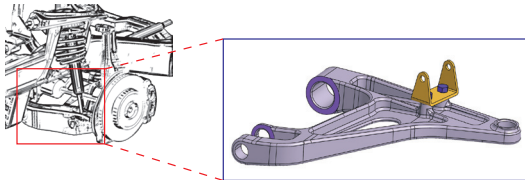


Figure 1: Suspension system assembly and control arm

Automotive manufacturers are usually working simultaneously with many projects and hundreds of analyses. Therefore, parametric models and analysis templates are valuable tools for evaluating different control arm models with minimal effort.

In the first part of this study, the stresses in a given control arm design are evaluated for several different cutout pocket radii. Typically, many load cases must be considered during design. For simplicity only two are considered in this example: a pothole load case and a vertical load case.

In the second part of the study, the original analysis model is used as an analysis template to evaluate a control arm from a completely different suspension sys-

Key Abaqus for CATIA V5 Features and Benefits

- Abaqus-based analysis within the CATIA V5 environment
- Nonlinear effects included, such as material plasticity and surface contact
- Parameterized geometry and analysis attributes within the same tool
- Ability to reuse analysis templates for multiple geometries

tem design. Using the analysis template approach, it is quite straightforward to replace the first design with another design from a different vehicle (Figure 2).

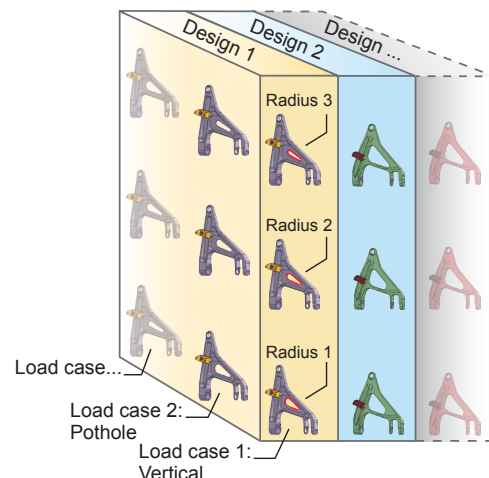


Figure 2: Using design tables and analysis templates to analyze multiple designs under various load cases

Model Description

The model consists of four geometric parts: the control arm, the shock absorber bracket, and two bolts. The control arm connects the steering knuckle to the chassis and allows the knuckle to pivot on a lower ball joint. The shock absorber, which controls vertical movement, connects to the control arm through the bracket (Figure 3).

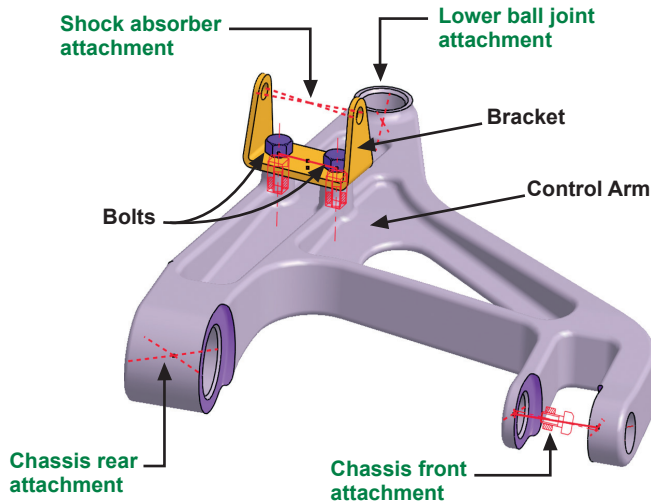


Figure 3: Control arm assembly

Constitutive Behavior

The material properties for the control arm include metal plasticity to provide accurate results if the yield point is exceeded.

Rubber bushings are used for both the front and rear attachments to the chassis (Figure 4). The rear bushing is modeled using a linear spring in the axial direction and a nonlinear spring in the radial direction to simulate the bushing's carefully tuned behavior (Figure 5).



Figure 4: Bushings

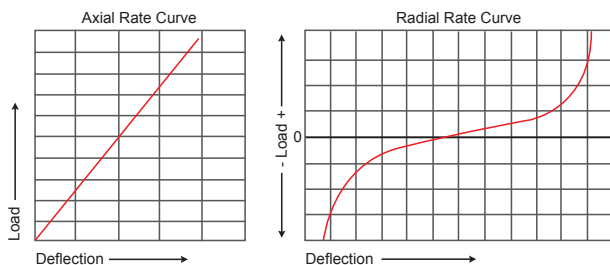


Figure 5: Bushing axial and radial behavior

Bolt Pre-Tensioning Effects

The two bolts connect the bracket to the control arm. The stress in the control arm at the location of this connection is particularly important. Therefore, the effects of pretensioning the bolts using bolt-tightening connections must be modeled accurately.

Surface Contact

Defining contact properties between two surfaces helps ensure accurate results. In this analysis, the contact between the bracket and the control arm is modeled without considering friction effects.

Approach 1: Performing a Parametric Study Using a Design Table

Through a parametric study, stress results can be evaluated for a number of design variations. A design table can be used to manage the model parameters and to switch quickly from one model configuration to another.

Load Case Parameters

As part of the design process, the suspension assembly model is subjected to a variety of load cases, such as braking, accelerating, and driving over a pothole. In this study, only pothole and vertical load cases are considered. These load cases consist of forces and moments applied at the chassis and at the lower ball joint's attachment points. Figure 6 shows the load locations and values for both cases. In total, fifteen load case parameters are implemented in the analysis: nine force components and six moment components.

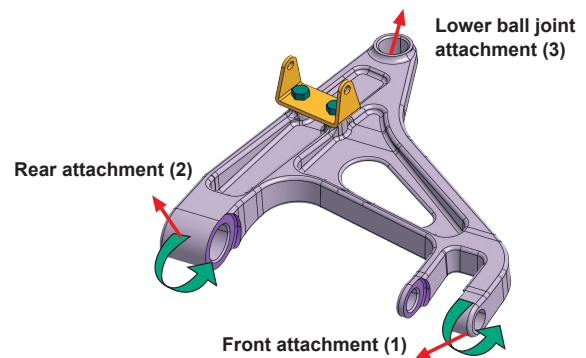


Figure 6a: Load and moment locations

Point	Loads (N)			Moments (N.m)		
	X	Y	Z	X	Y	Z
(1)	30	8400	-2500	24	-0.55	-0.6
(2)	-340	-13000	14000	27	-0.78	-1.2
(3)	10650	3700	21500	0	0	0

Figure 6b: Pothole load values

Point	Loads (N)			Moments (N.m)		
	X	Y	Z	X	Y	Z
(1)	50	5600	-3200	24.3	-1	0.3
(2)	-260	4350	15500	27	-1.1	0.075
(3)	550	-10500	16700	0	0	0

Figure 6c: Vertical load values

Geometric Parameters

The size of the control arm pocket can be a significant factor in the design weight and cost. Therefore, the effect of variations in the pocket dimension on the stress results is studied. Three pocket radii are evaluated: 12.5 mm, 10 mm, and 7 mm (Figure 7).

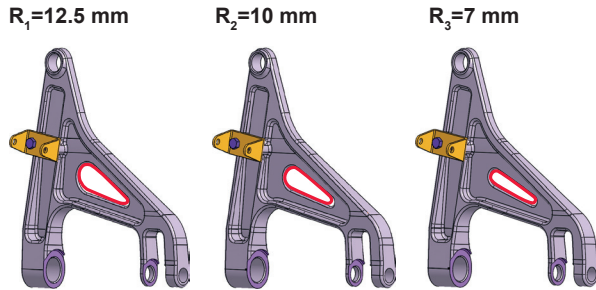


Figure 7: Pocket radii evaluated in the parametric study

Controlling Parameters through a Design Table

A design table provides a tool to create and manage sets of model parameters. Design tables drive the parameters of a CATIA based model from external values. These values are stored in either a Microsoft Excel file or a tabulated text file. Each column in the table corresponds to a model parameter, and each row contains a set of parameters to be considered together as a group. These sets are called configurations (Figure 8).

DesignTable, configuration row : 1							
Line	Config.	Arm/Pocket_Radius	FB_Fx	FB_Fy	FB_Fz	FB_Mx	FB_My
<1>	VerticalLoad+R1	12.5mm	50N	5600N	-3200N	24.3Nxm	-1Nxm
2	PotHole+R1	12.5mm	30N	8400N	-2500N	24Nxm	-1Nxm
3	VerticalLoad+R2	10mm	50N	5600N	-3200N	24.3Nxm	-1Nxm
4	PotHole+R2	10mm	30N	8400N	-2500N	24Nxm	-1Nxm
5	VerticalLoad+R3	7mm	50N	5600N	-3200N	24.3Nxm	-1Nxm
6	PotHole+R3	7mm	30N	8400N	-2500N	24Nxm	-1Nxm

Figure 8: Design table

In this study six configurations are defined: each pocket radius value is tested against each of the two load cases. Each time a new configuration is selected, the model is updated automatically. The design engineer steps through the design table to obtain the results needed

for each configuration (Figure 9). Alternatively, a Visual Basic script can be used to cycle through the various design configurations and run the analyses.

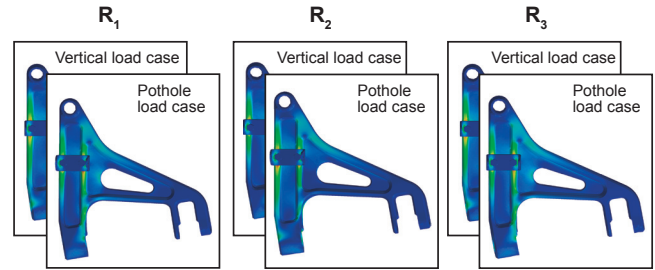


Figure 9: Von Mises stress results

Approach 2: Reusing Analysis Models with Analysis Templates

The last step of this study reuses the analysis model with geometry from another vehicle (Figure 10). The same stress evaluations can be conducted on different vehicle programs, allowing for great gains in efficiency with little effort.

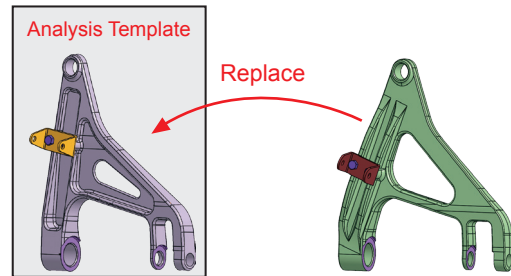


Figure 10: Different geometry evaluated using the same analysis template

The CATIA publication capability is used to create an analysis template. Using this approach, a geometric entity (face, edge, point, etc.) is associated with a unique and descriptive name called a publication. For example, the lower ball joint attachment surface is associated with the "LowerBallJoint" publication. All analysis attributes, such as loads and constraints, are applied to publications rather than directly to the underlying geometry. Bolt-tightening connections, springs, contact definitions, and loads are created based on publications. The ability to abstract the analysis attributes from the geometry is a fundamental requirement for creating analysis templates.

Figures 11 and 12 illustrate the difference between the standard approach (without publications) and the analysis template approach (with publications). Using the standard approach, analysis attributes are associated directly with the geometry. However, using the analysis template approach, analysis attributes are instead applied to publications.

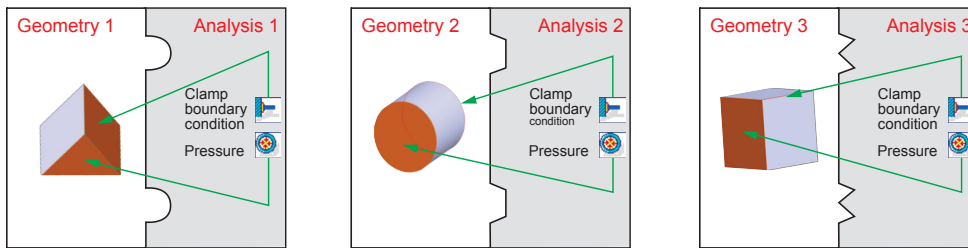


Figure 11: Standard approach. Analysis attributes are applied to the geometry.

Once an analysis is defined, it can be used as a template for other design evaluations. In this case the control arm and the bracket are replaced by a new design, while the analysis model and the design table remain the same. All analysis attributes update automatically

to associate with the new geometry. Then, each analysis in the design table is rerun for the new suspension system design. Figure 13 shows the von Mises stress results for the second control arm model.

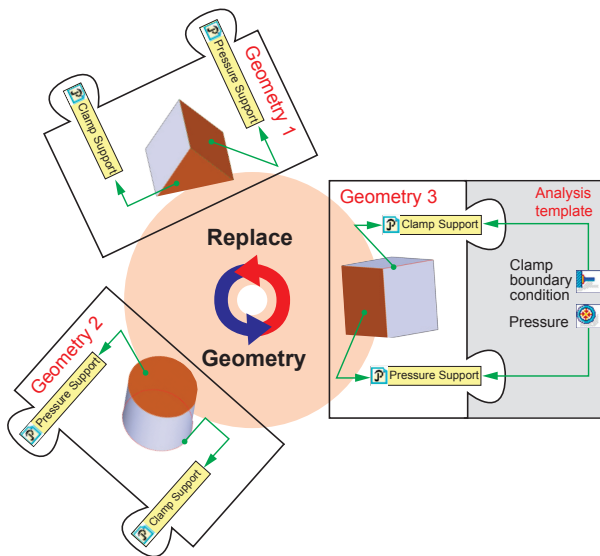


Figure 12: Analysis template approach. Publication objects are generated from geometric entities, and analysis attributes are applied to the publications.

The analysis template approach has a great advantage over the standard approach for analyses that will be performed on a number of different geometries. Various designs can be tested using the same analysis model, provided that each design uses the same publication names. Applying analysis attributes to publications instead of the geometry enables the modification or replacement of the geometry while maintaining full associativity between analysis attributes and geometry.

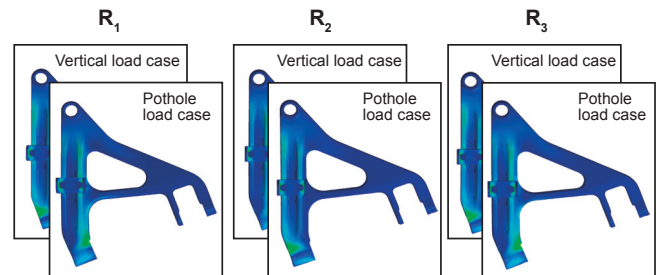


Figure 13: Results from the second control arm design.

The results for the two models can be easily compared to determine whether the design configurations meet the appropriate stress requirements.

Conclusion

This case study demonstrates that Abaqus for CATIA V5 can help to improve productivity during the design process. AFC provides the capabilities necessary to perform analyses easily and obtain accurate results, including nonlinear effects. The application also enables design engineers to iterate efficiently on an analysis, using tools such as design tables and publications, resulting in important time savings. If an alternate design must be considered, it is easy to use an analysis template with an updated design to obtain the necessary validation of requirements.

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 an open PLM platform 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, the SIMULIA brand provides sales, services, and support through a global network of regional offices and distributors. For more information, visit www.simulia.com.

The 3DS logo, SIMULIA, and Abaqus are trademarks or registered trademarks of Dassault Systèmes or its subsidiaries. Other company, product, and service names may be trademarks or service marks of their respective owners. Copyright Dassault Systèmes, 2008.

