ABAQUS as an Integral Part of Dana’s Engineering Strategy

Frank Popielas
Dana Corporation, USA

Abstract: In recent years in the automotive and heavy-duty industry the requirements for emission and cost reduction are becoming tougher. In order to fulfill these requirements and be a leader on the market CAE is the essential feature. Not only it guarantees a shorter development cycle but also general cost reduction. Cost reduction can be achieved through cycle time reduction, less material usage, optimal manufacturing setup, etc. At the same time the function of the product has to be maintained.

In order to achieve this a vertical integrated engineering approach is being developed and implemented in Dana for which CAE is the basis. In a large corporation like Dana this has to be done for a variety of products and implemented globally. This alone and the need for an optimal asset utilization requires standardization for a wide range of applications. On the CAE side we believe that such a standard should be developed around a structural code with sound links to all other major CAE packages and functionality as well as CAD and data handling.

For several years now Dana and ABAQUS have a corporate agreement. We took this time to define a communication structure between both companies and update in major divisions the CAE approaches (similar to the "unified FEA" from ABAQUS) by implementing state-of-the-art features into the code in order to be the leader in the market. The positive experience and the definite leadership of ABAQUS itself especially in regards to non-linearity and contacts were the basis to define ABAQUS earlier this year as the standard FEA code for the whole Dana Corporation. This includes the forward-looking strategy as formulated around SIMULIA. Thus, it is in the center our strategic CAE discussion for building an integral engineering approach with CAE right in the middle of it.

This paper discusses more in detail Dana’s approach and how ABAQUS plays a major role in it and how it is related to the market needs. Further it will be shown with examples how ABAQUS was integrated for some of Dana’s core products.

Keywords: CAE, CFD, NVH, Non-linear, linear, durability, FSI, Fluid Structure Interaction, Unified FEA, CAE Approach, SIMULIA, contacts, gasket elements, damping
1. Introduction

1.1 Dana products and place in the industry

Founded in 1904 by Clarence Spicer and Charles Dana the Dana Corp became over the years a company with a wide product portfolio serving as a major supplier to both the automotive and heavy-duty industry. The footprint of Dana is quite global. Dana is present in over 30 countries worldwide and supplies all major OE manufactures. The focus for Dana is mainly at this point on products “under the vehicle” and “under the hood”, which we call our core products (Figure 1). Those core products consist of axle, driveshafts, engine, frame, chassis and transmission technologies. With these products we are a leading and essential supplier to the automotive, commercial vehicle and off-highway industry covering more than 60 million vehicles annually. Our products are associated with well-known brand names like:

- Glacier Vandervell™
- Long®
- Nakate®
- Plumley®
- Spicer®
- Victor Reinz®.

Naturally a wide range of engineering is involved on order to provide high-quality products starting from product engineering through testing up to manufacturing. Manufacturing includes traditional techniques like cutting, stamping, welding (from traditional spot welding up to laser welding), forging, heat treatment, screen printing, etc. In order to stay on top we also look into new technologies like hydro-forming, AtmoPlas™, etc.

The products themselves undergo a constant evaluation in order to provide the best application technologies. Thus, new technologies are being developed on an ongoing basis – electronic torque coupling, Intelligent Cooling™, WaveStopper™ for Multilayer-Steel gaskets, Multi-layer acoustical protective shields to improved heat and noise shielding to name a few. In order to achieve this part of the research within Dana includes material research. This starts from understanding of the best materials available on the market and tune it to our needs up to in-house developed proprietary material like the different coatings and rubbers used for gaskets and valve stem seals.

Major drivers in the development are emissions, NVH, cost and to some extend for our product design itself. What their impact on engineering is will be discussed later in this paper.

This paper tries to show how Dana takes advantage of available engineering technology with focus on CAE in particular why CAE plays in our view the major role. Combining this it will be illustrated how Dana is utilizing ABAQUS as a leading FEA code and what we believe the future direction for CAE will be and why ABAQUS plays for Dana a major role.
As mentioned earlier major drivers for our products are emissions, NVH, strength/durability, cost and design. This of-course is not only true for Dana’s products but also for the products in our industry in general. Let’s discuss this more in detail:

**Costs**

In general, it is straight forward that our customer want to have for the same price a better product or for the same product a better price over the years. There are several ways to achieve this:

- Less material usage ⇒ lighter structures,
- Less weight ⇒ lighter structure or different materials,
- More efficient manufacturing process,
- Reduced development times,
- Better quality.
Emissions

Emissions are a result of the interaction of the whole system, but some general factors to control emissions are:

- Less weight ⇒ results in less fuel consumption,
- Reduced engine component distortion:
  - Cylinder head and block ⇒ less oil consumption,
  - Exhaust system ⇒ better sealing potential especially in high temperature areas,
- Optimized engine calibration, this includes:
  - Firing pressure,
  - Injection pressure,
  - Cooling system,
  - EGR,
  - Fuel,
  - Etc.

NVH

One of the major focuses nowadays is mainly NVH (Noise, Vibration, and Harshness) with emphasis on the noise component. Specifically we have to address the question of what can be done to reduce the noise/sound for the passengers in a vehicle and for the surrounding environment? This is why we are investigating new design elements, damping characteristics, vibration profiles (based on design, material, and manufacturing process), and advanced shielding of noise source.

Strength/Durability

Quality guarantee of products doesn’t only imply that a component is functioning under specific operating conditions alone. It is also an assurance that over the lifetime of the system, under maximum conditions, the product functions without problems. Based on these requirements, emphasis is paid towards fatigue and durability simulations and limit studies.

Design

Visual look and appeal is an important factor, which is mainly driven by how the market is accepting a product. For CAE, though it is not the main factor, engineers have to make sure that the design/look is not interfering with the functionality of the product. It is just another constraint.

What is now the relevance of CAE?
As soon as we think about reducing development time CAE should come into mind. Experiments can be simplified, controlled, or accelerated up to a certain limit. However, with the advances in computer technology and software capabilities, we are able to simulate very detailed the conditions as we see them in the application of our products. The bottom line is, CAE helps to reduce time and delivers results, where the experiment is not capable. Furthermore, simulation techniques allow studying design features, comparing materials, assembly techniques, etc… We use it for theoretical DOE, max/min studies to ensure that the proposed design is functional within the specified production tolerance, and we use it to define those production tolerances.

It is important to maintain material databases and other input data needed in order to accurately solve the problem before hand. Just having the knowledge of how the software works and having the best hardware doesn’t do the trick. Thus, in order to trust the simulated results, simulation and experimental results need to be correlated.

It is difficult to explain in a few words the general importance of CAE for R&D. The focus of this paper is to provide guidance from the higher level – the product in general, down to the detailed level of designing and manufacturing. Looking through the industry, we see that the leading companies have adopted CAE techniques into their production and development routines successfully. The implementation of CAE for design simulation was the first step. It started very simplified about 20 years ago. Again, with the advances in software and computer technology we can simulate now very detailed models in a short time. The next step was to simulate certain aspects of the manufacturing process, system application, material properties, etc…, however, all those simulations were still island-like. Today the leaders are approaching new levels of simulation techniques, which will be covered later in this paper. Steps 1 and 2 are common industry standards. Step 3 is being developed and implemented partially by the leaders in the industry.

2. **CAE implementation in Dana and it's focus on ABAQUS**

Dana recognized at an early stage the importance of CAE. Being a “just” a supplier focusing on the simple traditional (let’s call it hand-on) product engineering, we went through the full system supplier stage with focus on product FEA only. Our goal is to be a development partner to our customer, which means we have to connect the different engineering/CAE-islands into a system (Figure 2). If we would go into more detail of this chart we could find all mentioned above drivers (paragraph 1.2.) again present here.

In order to be able to simulate a system, it is necessary first to understand the components of the system, starting from manufacturing of the product. The next step is the understanding of how those modules are linked with each other, understanding the sequence of events. Linking of modules does not only mean we know the sequence, it means we know what information needs to be communicated from one simulation stage to the next. Such information is usually limited to residual stress, strain and shape. However, reliability studies require durability simulation of the part and system as well. Therefore, it would be beneficial to know the history of the product, meaning what damage occurred to the part throughout manufacturing and actual life.
Figure 3 shows the CAE approach for thermal-acoustical protective shields as an example of such linking of different modules.

The goal is to achieve wide implementation of our simulation techniques, standards have to be developed and well defined, such as workflow, pre and post processing techniques, naming conventions, etc… This will allow pushing the most repeatable tasks, which can be automated almost 100%, down to the product and design engineer levels. Those tasks are meant to run at product and design engineer levels at the earliest development stage, which allows most of the optimization work to be done before it goes into a complete system analysis at sophisticated level. Dana implemented in R&D such techniques up to a certain level already to up-front identify manufacturing capability of designs, cost effective setups/material usage, and, of-course, for quoting purpose.

The CAE analyst’s responsibility is to:

- Develop simulation techniques, which includes:
  - Update and develop for the product appropriate CAE approaches;
  - Software evaluation and development,
  - Develop and generate the necessary input data, if needed experimentally,
  - Result correlation with experimental data,
  - Develop automation techniques to speed up turn-around;
• Define simulation standards for the day-to-day tasks, which can be pushed out to product and design engineer level;

• Run the more sophisticated simulations, which require enhanced CAE knowledge in regards to theory, software and hardware understanding.

Based on Dana’s product range we are looking into the following types of simulation and outputs, as examples:

• Structural simulation:
  o Material stress and strain,
  o Shape,
  o Frequency response,
    ▪ Modal analysis, and
    ▪ Harmonic analysis,
  o Fatigue,

• Thermal analysis:
  o Conductivity, and
  o Convection,

• NVH:
  o Sound power, and
  o Radiation efficiency;

• Fluid domain simulation:
  o Fluid speed,
  o Fluid flow efficiency/distribution,
  o Air- and oil separation efficiency,
  o Engine coolant circuit simulation in order to generate thermal map for structural simulations (steady state and transient),

• Plastic and rubber molding

• Tool design for forming dies

• System simulations:
Part assembly (in general simulation techniques as mentioned above for structural simulation),

Fluid structure interaction (FSI) – interaction/link between structural and fluid domain.

Crank and cam shaft alignment and interference with attached parts.

The tasks we are faced with are ranging from:

- **Linear analysis, for some of the products like:**
  - Axles,
  - Driveshafts,
  - Frames,

- **Non-linear analysis, for products like:**
  - Axle systems and subsystems,
  - Cam and valve covers,
  - Cooling systems,
  - Driveshaft assemblies and joints,
  - Flat gaskets (cylinder head gasket, exhaust manifold gasket),
  - Fluid routing,
  - Frames,
  - Fuel cell components
  - General engine system simulation
  - Radiators,
  - Rubber gaskets (for example for valve covers)
  - Thermal-acoustical protective shields,
  - Valve stem seals.
Non-linear analysis is the dominant analysis type. The model size or DOF depends on the used element types. Models have today around 1M to 5M DOF. Getting into more detailed engine distortion simulation, for example, where we would have to use second order elements, we easily see over 10M DOF, and even more in the near future.

Besides those issues we have to manage major contact simulations based on the nature of our products and the environment they are operating in:

Figure 3: CAE Approach for thermal-acoustical protective shields
Multi-layer steel gasket, when explicitly modeled in 2D and 3D due to the interaction between the layers and the components during manufacturing and operating conditions in engine;

Multi-layer steel gaskets using gasket elements for the 3D simulation of engine hardware as global model;

Thermal-acoustical protective shield due to its multi-layered sandwich structure during manufacturing and operating conditions;

Valve and cam cover simulation with its interaction with rubber gasket and cylinder head;

Crash analysis of frame structures;

Assembly simulations for axles and drive shafts;

Valve stem seal cyclic simulation;

Fluid structure interaction for:
  o Fluid and thermal products;
  o Engine exhaust system;
  o Valve stem seals to define up-front the oil metering.

Over the years Dana had experience with different software packages. Although, there is a variety of different products Dana recognized the need to standardize on main software packages, covering the main simulation needs. We see the structural component of the simulation in our business as the center of our simulation efforts. Evaluating Dana’s product requirements and available on the market major software packages, Dana introduced a couple years ago at two major divisions (Sealing Products and Torque Traction) ABAQUS as the standard software package. This recognizes ABAQUS as the leader in FEA solving techniques, and development partner, helping Dana to improve simulation techniques, and improved accuracy of our results.

Based on this positive experience, Dana decided at the beginning of 2006 to implement ABAQUS as the main FEA package throughout the corporation. We see this as a major step forward to better utilize our analyst’s experience, CAE system setup, in regards hardware and automation throughout the corporation, despite the differences in products.

3. Examples for a few products

This paragraph provides a brief overview on how we use ABAQUS for our products. For some we are in process of implementing ABAQUS as we only recently switched there or added the
structural simulation as an additional module to the CAE approach through fluid structure interaction (FSI).

3.1 Axle and drive shafts

In our Driveshaft Engineering department, we have fully leveraged the use of ABAQUS, Python, and ABAQUS Viewer’s Plug-In capability to automate the analysis of cardan joints, the core of our driveshaft product. Consequently, the CAE analysts spend just minutes of their time going from a completed mesh to final results suitable for publication in a report. The results are formatted in the desired view with dynamic leader flags attached to the critical stress areas, image files are create, and tables containing stress results are written to summary files. Quality assurance checks like stress convergence are done automatically within the process and are recorded as well. (see Figure 4)

![Figure 4: Cardan joint](image)

For our light axle product line, we have utilized the non-linear strengths of ABAQUS to run full axle assemblies that contain every component from the axle input to the output at the wheel. The use of surface based contact, pre-tensioned bolted connections, and press fits has allowed us to virtually build and test an axle assembly and run multiple tests simultaneously. While the initial set-up is longer than past individual point FEA runs, the overall time to run all the virtual tests has
been reduced since they are all “tested” at once. In addition, we have improved results for each of the virtual tests like assembly stress, fatigue, leak path analysis, and gear deflection since we have reduced assumptions and improved boundary conditions through the use of full assemblies. (See Figure 5)

3.2 Multi-Layer Steel Cylinder Head Gasket (MLS CHG)

Following a CAE approach (Figure 6) ensures that we cover all the main parameters of the system “engine/gasket”. It also provides us with the opportunity to optimize the gasket up-front to address the engine requirements.

The approach starts with manufacturing of the gasket layers – beading and applying the coating. This is covered in the single layer analysis module. The next step is assembling the layers to a multi-layer stack together. Two-dimensional axis-symmetric analysis is the main approach to detailed problem. This type of analysis provides large amount of information on the behavior and interaction between the engine hardware and also within the different layers of the gaskets as well. A typical 2-D axis-symmetric multi-layer model consists of the different layers of gasket geometry depending on the design. Single layer analysis results are transferred to include the mesh for the
metal layers, mesh for the rubber coating on the metal layers and also the residual stress in the material created during the forming process. The previously in-house developed simulation code Proteus® was in the process combined with ABAQUS to utilize better ABAQUS’ scripting features and development resources for future enhancements. This is now known under Proteus®-powered-by-ABAQUS.

2D axis-symmetric model is also used to estimate the durability of the whole gasket under operating conditions. The gasket is compressed to an operating thickness and then it is cycled with a small displacement called head lift off. This value is usually measured using experimental tests. During the cycling of the gasket, damage is accumulated and it eventually fails. Specialized Python scripts have been developed to estimate the durability/damage during the forming process and add it to the damage accumulated during the cycling process.

The 3D simulation utilizes the whole engine hardware or in case of a V-engine structure one half most of the time. This simulation uses ABAQUS gasket elements to represent the cylinder head gasket assembled with a deformable solid FEA model of the head and block and other structures to simulate the sealing performance of the entire MLS CHG in the engine assembly. At the end of the simulation, specialized Python scripts are used to create a complete report consisting of sealing pressure plots, comparison charts and gasket thickness tables which the analyst use for further

Figure 6: CAE approach for MLS CHG
optimization of the gasket and also for correlation studies. Several engine performance characteristics and behavior are also studied from this simulation. (Figure 6 and Figure 7)

As mentioned above the whole system becomes more and more important as a whole. That’s why more often besides head and block also other components like the exhaust manifold are part of the simulation model as they strongly interact with each other, especially under thermal cycling conditions (Figure 7). The thermal information, like coolant circuit or exhaust gas flow usually is initially generated through CFD simulations (Figure 6 and Figure 7) In our case we use Fluent as the code. Special developed Python scripts transfer the information from the fluid domain onto the structural ABAQUS model. In some cases, especially for exhaust system simulation we use MpCCI to investigate the interaction between both domains utilizing FSI.

Most of the simulation is done in \Standard. Delicate forming simulation in 2D or 3D are or performed completely in \Explicit or a mix between \Standard and \Explicit.

![Figure 7: MLS CHG and EMG system analysis](image)

### 3.3 Thermal-acoustical protective shields (TAPS)

The CAE approach for TAPS (Figure 8) helps in using multi-tasking approach to make this product to the prototype faster than ever before. Once the CAD designer creates the initial design based on the thermal, acoustical and space requirements, the design is evaluated for forming,
minimum Eigen-frequency at the same time this helps the CAD designed to further fine-tune his design before it reaches an analyst.

Python scripts are used to integrate CAE into an automated workflow, by bringing together technologies like Apache, Sun Grid Engine, ABAQUS Standard and Viewer, FTP, email, etc. CAD designers have a web based resource for submitting ABAQUS jobs on remote systems. A Python based job submission form provides the designer with appropriate choices for the simulation. Upon job submission, Python is used to assemble ABAQUS input decks and submit them to a queuing system. Customized ABAQUS environment files are used to start simulations, move files to central storage spaces, and execute post processing applications. In systems like these, ABAQUS Viewer is completely automated for post processing, no user intervention is required.

The initial forming process is One Step forming simulation, which is a reverse forming approach (to manufacturing feasibility of the shield). Once the geometry from the CAD designer reaches the analyst, the next step is to generate the forming die for further detailed analysis using die designer software. The next step is incremental analysis. The incremental analysis uses ABAQUS Explicit in which the different layers of the TAPS are represented by a shell element layer, and the tool consists of rigid elements. The results from this simulation provide detailed understanding regarding wrinkling, thinning and high strain areas.

The structural dynamic analysis uses the input from the forming analysis module -formed shape, thickness, stresses, and strain. This information is used in the harmonic analysis of the shield, which in turns predicts the high stress areas during the operation of the shield. A specialized Fortran code is used at the end of the harmonic analysis to predict the possible area of cracking on the shield due to the operation.

![Figure 8: CAE approach for TAPS](image)
3.4 Covers and rubber gaskets

The need to reduce weight and cost has lead to an increased use of short fiber reinforced thermoplastic or thermoset plastic cam covers. These isolated engine cam covers are part of complex structural assemblies, composed of the covers themselves, elastomeric gaskets and grommets, limiters and bolts, metallic or plastic baffle plates, connectors for electronic controls and Air/Oil separation elements. Dana Corporation uses ABAQUS in the design, verification and optimization of each component and, also, the whole assembly.

The primary function of the cover is to provide an oil leak proof enclosure to the cam system, while maintaining structural integrity. In addition to this, cam covers have been identified as a major contributor to the total engine radiated sound power, major efforts are directed to improve the transmissibility of these components. The applications of ABAQUS to the analysis of a cylinder head cam cover assembly range from 2D to 3D, from static to dynamic calculations.

2D Static Analysis - Preliminary to the actual 3D FE calculation a vertical tolerance stack up analysis has to be conducted to determine the extreme dimensions of the components that will result on the minimum and maximum compression of the gasket and grommet. Given the non-linear nature of the elastomeric materials response, evaluation of the minimum sealing pressure and maximum stresses in the component at these extreme dimensions are perhaps more important than the nominal configuration.

The elastomeric materials are modeled as hyperelastic. The more rigid cover and block head are modeled as rigid surfaces. Contact between the elastomeric components and the cover or is endowed of Coulombian friction.

The results from the simple 2D calculations for the gasket and grommet are combined into a so-called load balance analysis. This information is very important as it can be used to determine the equilibrium stand-off, the maximum and average sealing pressures, contact widths, maximum stresses in the elastomeric components, etc. The optimization of the isolated system is done, as a first approximation, at this stage of the design.

3D Static Analysis - A three dimensional analysis of the complete assembly is necessary in order to evaluate its structural integrity and the sealability of the system. The analysis simulates the mating of the components at room temperature, followed by temperature excursions up to 150 °C and -40 °C. The following parameters can be evaluated: a) cover stiffness, in particular flange deflection and maximum stresses, b) bolt pattern and spacing, c) maximum estimated creep, d) out of flatness, e) presence of steps, etc.

The finite element of the cover is complete modeled using second order modified tetrahedral elements. The elastomeric gasket and grommets are modeled using like gasket elements, which force response is determined directly from the 2D finite element analysis. Compressing the top of the grommets by an analytical rigid surface while the bottom of the gaskets is supported by a fully restrained rigid surface imposes the loading on the system. The only simplification made to the analysis is to assume the cover is isotropic, thus neglecting the anisotropy of the material due to fiber orientation and residual stresses introduced during the molding process.
The finite element predicted stand-off is compared, at this point, with measurements conducted on actual components. An important result is the average sealing pressure produce by the gasket, as this type of information can be used to anticipate problematic (low sealing pressure) spots early on in the design process.

**3D Dynamic Analysis** - The steady state dynamic analysis is conducted on the same cam cover model as the one used for the static analysis. An important aspect of this type of analysis relates to the frequency-dependent viscoelastic material properties. The storage and shear moduli are determined by conducting tests in a Dynamic Mechanical Analyzer (DMA) machine, using either a single cantilever or a shear-sandwich configuration.

The information gained from this analysis is compared with experimental results and directly used to select the gasket and grommet material and/or the geometry of the cover self in order to modify the dynamic response of the system.

**Figure 9: CAE approach for covers and rubber gaskets**

### 3.5 Frame structures

In the vehicle architecture group (Structural Solutions), we would explore ABAQUS explicit nonlinear analyses as an alternative to ESI Pamcrash and LS Dyna for crash energy absorption design. We would introduce these analyses in light of BMW’s acceptance of ABAQUS for vehicle crash analyses. Analyses also performed in our group include durability, forced vibration response and
Normal Modes analysis. Also our fabrication group performs implicit and explicit component metal form analyses, including tube hydroforming.

Figure 10: Full underbody frame for Sport Utility Vehicle

Figure 11: Using ABAQUS as a primary solver for front recovery loop (tow pull) non-linear analyses. A Von Mises stress contour is shown
3.6 Fluid and thermal products

At Thermal Products, we offer advanced thermal management solutions for light-vehicle, heavy truck and small engine markets. The Advanced Engineering Center is responsible for the development of components and subsystems in support of typical automotive power plant thermal management systems and fuel cell balance-of-plant (BOP) requirements and fuel processor technology. Heat Exchangers, pre-heaters and complete thermal management systems are being developed by CAE. Development and customer support activities involving FEA stress analysis include: heat exchanger pressure loading, heat exchanger and bracket vibration analysis and thermal stress modeling.

The vibration analysis of a typical radiator assembly is shown in Figure 13, the complete analysis includes a frequency scan, resonance mode shape analysis and harmonic response stress analysis all in an effort to design brackets that can withstand the challenges of off-highway applications. Figure 14 is an example of a stress analysis on a high temperature heat exchanger used as part of a reformer based fuel processor for hydrogen generation for fuel cell engines. The reformer consists of a series of catalyst reactors and heat exchangers that are used to crack hydrocarbon fuels to generate hydrogen. Because of the elevated temperatures, the thermal gradients and therefore high stress levels are of concern to system durability. Our product innovation focus of the future is a higher level of CAE integration including fluid structure interaction, which will involve coupling our CFD simulation capabilities using Fluent to the structure stress analysis efforts to provide thermal management products of the highest performance standards.
Frequency scan
Resonance mode shape investigation
Harmonic response stress analysis

Radiator mounted on shaker table

Figure 13: Radiator vibration analysis

Temperature distribution
Stress distribution

Finite element model

High stress area on the hot end header

Figure 14: Reformer cooler thermal stress analysis
3.7 Valve stem seals

Valve stem seals are even though small parts a challenging object. For a general structural analysis, i.e. internal material stress under certain loading (force and thermal load) a 2D axi-symmetric simulation in ABAQUS \Explicit is performed. Here the part is analyzed for stress and strain to ensure we don’t exceed material limits. Special experiments were performed to describe the hyper-elastic behavior of the rubber seal as accurate as possible.

The main challenge for this type of seal though is the prediction of oil metering through the contact between seal and stem, which is almost 0 to begin with. It is know that an oil film changes the pressure onto the seal and, thus, the internal stress of the material. Furthermore, time and motion frequency of the stem do their part as well in changing the pressure of the seal onto the stem on one side and material properties on the other. Here it is necessary to take full advantage of Fluid Structure Interaction. Coupling two codes (ABAQUS \Explicit and FlowVision) solved this problem. The control of both codes and transferring the data between them are provided by Capvidia’s Multi-Physics Manager. (Figure 15 and Figure 16)

![Figure 15: FSI for Valve stel seals](image_url)
3.8 Correlation

To begin with it was mentioned that without experimental support this system would not work. Only the knowledge of the software and the right hardware setup doesn’t guarantee good results. Major emphasis has to be on correlating the simulation results. That is why part of establishing the right CAE approach for the product has to accompanied by defining the parameters to be used for correlation.

Each product group established those parameters before releasing the approach to production. Especially, before pushing simulation techniques down to the product and design engineer level one has to make sure that this correlation has been done. Once released for 100% to the production level, the engineer has to rely on the results for sure, in order to feel confident in the decisions based on simulation results. This doesn’t mean that the software has to be able to cover every eventuality. It simple means that the limits for the particular application are well known.

Some very general correlation parameters are:

- Stress and strain, which can be correlated through hardness measurements for metals;
- Shape;
- Thermal information, and
- Fatigue.

It sounds simple, the difficulty is to have the correct input data. For that, a good experimental basis is of importance. The bottom line is, in order to have good results, one has to understand in every detail the product and process to be simulated and how it can be simplified, i.e. which are the controlling factors.
4. Dana’s CAE strategy

It is clearly understood that in order to achieve good business, sustain markets and develop those further a vertical integrated engineering approach plays a major role. In this CAE carries nowadays the main load, as it is the main part for developing and designing parts and processes within a short time, and cost effective. Dana is full aware of this and started years ago to pay attention, and invest into this.

What started with a product focus, developed soon into the development of CAE approaches. At the same time Dana started actively to get into ABAQUS as the main FEA code as the main focus was and is on structural analysis. Intensive knowledge exchange between both companies allowed implementing new features into the ABAQUS code to support Dana’s product and process simulations better and better.

Over the years, the first relatively simple CAE approaches developed into more complex structures, which included modules covering other simulation techniques and, naturally software packages. What Dana calls “CAE approach” found its place in ABAQUS under “Unified FEA”. Focusing more and more on transferring simulation results from one simulation stage to the next was the first step. This means openness to other software packages in order to be able to exchange data without loss of information.

In a next step we started to look into sharing data between different simulation types on the fly, meaning that both simulation run simultaneously and influence each other results based on the increment they are in. ABAQUS started actively to develop FSI, which Dana actively was getting involved with, for instance for:

- Valve stem seal oil metering simulation connecting ABAQUS and FlowVision from Capvidea;
- Air and oil separation and it’s influence on structure using MpCCI to couple ABAQUS and Fluent;
- Exhaust system simulation to better control emission, again using MpCCI coupling ABAQUS and Fluent.

Another link Dana is using is transferring information between ABAQUS and Virtual.Lab from LMS for NVH simulation. Furthermore, in order to cover the “strength/durability”-driver we actively use the “ABAQUS – FE-Safe”-link. This is not only heavily used already but is being deployed to more application rapidly.

The important thing for Dana is that ABAQUS is not only a leader in FEA, but also started to team-up with other providers on order to provide a system, where information/results can be exchanged without loss. At the same time, significant advances were made in better covering simulation techniques, like forming, crash simulation, wear, parametric modeling capabilities, new element types and focus on ease-of-use in pre- and post processing, and performance overall, etc…

2006 ABAQUS Users’ Conference
Those capabilities are part of the reason why Dana started to standardize not only within product
groups but also across borders. System setup, automation techniques are easier to share now.
Meaning, the decision to go with ABAQUS was not just an administration driven decision but
mainly influenced by having technical advantages.

An interesting fact for Dana was the new ABAQUS being part of Daussult. With ABAQUS being
the driver for the SIMULIA initiative we see our CAE approaches expending to even more
integral approaches, covering the whole engineering process. Based on an open and common
exchange format platform we see the opportunity, effectively combining all engineering
disciplines from development, design, manufacturing to effective cost estimation or vice versa.
We believe that CAE stands right in the middle of it. ABAQUS is, therefore, a main partner for
Dana as we have similar understanding of where the development in simulation goes.

5. Conclusion

Dana developed extensive expertise in CAE over the last years. This acknowledges the importance
of CAE in our industry today. Being a leader in our field means we have to drive the development
and work with leaders together. ABAQUS represents for Dana such a partner.

The brief overview of how Dana is using CAE, and ABAQUS in particular, shows the depth of
theoretical simulation within Dana and our expertise. At the same time it shows our commitment
to develop new techniques and to drive the development.

6. References

   ABAQUS Users’ Conference, New Port, Rhode Island, USA, 1996
5. ; Bastias, P. C., Lu, C. and Anderson, M., “Applications of ABAQUS to the Design and
   Performance of Engine Cover Assemblies”, SAE 2002-01-0458
   SAE 2003-01-0483
17. A. Aksenov, et.al., “Influence of Interaction between Oil and Rubber on Valve Stem Seal Oil Leakage”, ABAQUS World User Conference 2006, Boston

7. Acknowledgement

I would like to thank for their contribution to this paper my colleagues: Pedro Bastias, Jim Carifo, Jim Cotton, Anthony DiGregorio, Tom Essi, Blake Garretson, and Rohit Ramkumar.