Comfort Evaluation of Foam Seats

Using Realistic Simulation

Finite Element Analysis (FEA) is accepted across a wide range of industries as an invaluable tool for product design and optimization. Design engineers from IDOM and Centro Tecnologico Grupo Copo are using Abaqus FEA from SIMULIA to evaluate the "feel," or comfort, of car seat foam—basically performing numerical modeling of tangible human criteria.

Cushions, backrests, headrests, armrests, and other foam parts that make up a vehicle seat are designed according to four principal criteria: integration within the vehicle, safety, aesthetics, and comfort. Satisfying all of these requirements, while working with complex foam material, makes the process of creating the seats complicated and time-consuming. In addition, measuring and evaluating results—particularly in the subjective area of comfort—presents a particular challenge to designers.

When designing car seats, most of the variables to be considered relate to either geometry or materials. Since the mechanical properties of foam are highly dependent on their strain level, multiple test specimens must be built to evaluate response and satisfy design requirements. Preparing a specimen for testing can take weeks: first, the mold with the desired geometry must be built, then the specimen is created in this mold, and finally, the specimen must be readied for the testing criteria being examined.

A valuable tool for facilitating and shortening this complex design process is numerical simulation using finite element analysis (FEA). Modeling the seats in a virtual environment integrates CAD with material databases and allows the input and evaluation of a variety of loads and stresses without the time constraints of reality testing. FEA can predict the response of a particular design under specific circumstances and supply data that can be used to optimize geometry and materials. But how can the simulation results be used to evaluate comfort?

One of the difficulties inherent to comfort assessment is to translate the sensation of comfort into quantifiable variables in order to measure comfort from the mechanical point of view.

Measuring seat comfort in vehicles can be carried out under both static and dynamic conditions. Typical variables for comfort assessment are related to occupant's position, such as hip-point or Seat Reference Point (SRP), to pressure distribution, or to the response of a seat specimen with an occupant model on it under the effect of a certain vibration spectrum. Variables related to comfort assessment can be obtained from CAD-geometry of the assembly vehicle-seat-occupant or from the mechanical response of the seat to certain tests. Since all these tests can be simulated numerically with FEA, comfort can be assessed in a design analysis environment.

At IDOM, we conducted a preliminary feasibility study in the framework of a more ambitious program aimed at developing a virtual environment for foam seat testing. This study focused on the assessment of numerical simulations, primarily related to comfort, using Abaqus FEA software from SIMULIA.

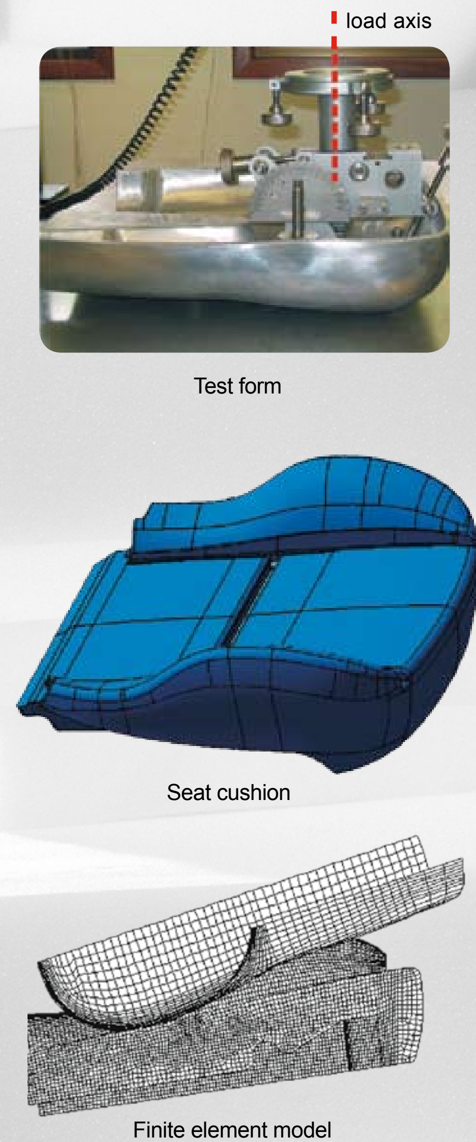

Figure 1: The top and middle images show the test form used for the indentation test and the geometry of a seat cushion. The bottom image shows the finite element discretization of both parts (test form on top of seat cushion) used to simulate the indentation test. The test form was discretized with 1642 4-node rigid elements (R3D4). The reference node of the surface of the test form lies on the connection between the test form and the load axis. At this node, all degrees of freedom, except vertical displacement, are restricted. The seat cushion was discretized with 8-node brick elements (C3D8). To reduce computational time, only half of the seat cushion was considered in the finite element model, taking advantage of symmetry with respect to a vertical plane. The mesh has 125,164 elements and 138,172 nodes.

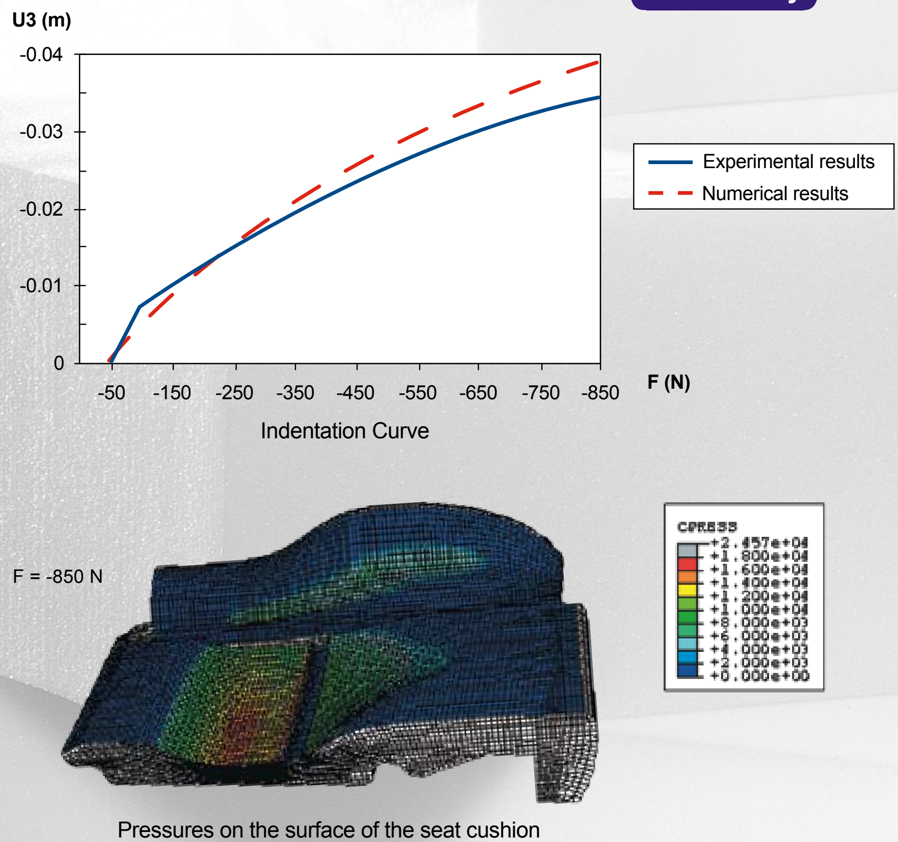

Figure 2: The top graphic shows the comparison of the indentation curve (vertical displacement U3 of the test form versus applied load F) obtained experimentally and numerically. The bottom image shows the contact pressure (Pa) on the upper surface of the seat cushion for an applied load of 850 N on the test form. Using an HP workstation with a 2.8 GHz IntelXeon processor and 1.5 Gb of RAM memory, the computational time required for the simulation of the indentation test with Abaqus/Explicit was 2 hours.

As part of our study, a physical test for static indentation was simulated numerically. The aim of this test is to simulate the mechanical response of a seat cushion when an occupant sits on it (Figure 1). Lab testing was carried out on a seat cushion positioned on a rigid support. A test form that reproduces the occupant's thighs was placed over the seat cushion. A vertical load was applied gradually on the test form, moving downward on the seat cushion to simulate the action of an occupant sitting on it. The penetration of the test form on the cushion was measured for certain loads applied to the test form.

In the general methodology of numerical simulation of the static indentation test on foam seats, two steps can be clearly differentiated: fitting of the general constitutive material model for elastomeric foams to the particular case considered, and simulation of the static indentation test itself.

Flexible polyurethane foams are hyperelastic materials. They have certain properties that make them very suitable to serve as seat cushions, because they significantly contribute to comfort. These foams can elastically undergo large strains, up to 90% in compression, and they have excellent energy absorption properties. They are highly nonlinear, they have viscoelastic properties, and they suffer from material softening in the first load cycles (Mullins effect).

The Ogden material model for highly compressible hyperelastic materials is contained in the Abaqus code. The three main hypotheses of this model are isotropy at a macroscopic scale, hyperelasticity, and non-hysteresis. The values of material parameters can be determined by means of a least squares fitting from stress-strain measures of simple experimental tests.

The material database used for this study contains a wide range of polyurethane foams, the specification of the polymer required to manufacture the foam, and different material and mechanical properties. The design process is significantly facilitated by means of this database, because of the capability of evaluating the performance of the same geometric design with different materials.

In our study, once the generic constitutive material model for elastomeric foams was particularized with experimental data of simple tests, more demanding tests, such as the static indentation test, were simulated. The curve-applied load—vertical displacement of the test form obtained in the simulation of the indentation test—showed good agreement with the one obtained in the experimental test (Figure 2).

Additional information relevant for comfort assessment can be easily obtained from such a simulation. Examples are contact pressure distribution on the upper surface of the seat cushion, vertical stress distribution in the seat cushion or contact area, and distribution of the load between seating plane and side wings.

As a consequence of the preliminary study, expectations about the potential of the application of numerical simulations with FEA in comfort assessment were met: experimental tests aimed to obtain results useful to compute comfort parameters can be successfully simulated numerically. Moreover, a great amount of information useful for comfort assessment can be easily obtained from these simulations.

The virtual testing approach also significantly reduces the time required for each design loop, allowing the design process to take advantage of the analysis to optimize the geometry in a virtual environment. FEA is definitely a valuable tool for assessing seat designs for comfort, and it will play a major role in this field in the future.

Natalia Camprubí and Fernando Rueda, Advanced Design & Analysis Division, IDOM

Iván Alonso, Centro Tecnológico Grupo Copo

For More Information
www.idom.es/ada
www.grupocopo.com