Comparison of different element types in structural analysis

K. Elsäßer*, B. Keding**, H. Müllerschön**, C. Pedrazzi*,

* TRW Automotive

TRW Occupant Restraint Systems GmbH&Co. KG

Industriestr. 20

73553 Alfdorf

** DYNAmore GmbH

Industriestr.2

70565 Stuttgart-Vaihingen

1. Introduction

Components for occupant restraint systems usually undergo structural analysis before hardware prototypes are made. In most cases the simulation is nonlinear in geometry, material and boundary conditions. Additionally, the load case often is highly dynamic. For this purpose explicit FEM is a suitable tool. It is well known, that the accuracy of deformations and especially stresses has to be checked carefully in some cases. Therefore, a comparison of different modeling approaches has been performed for basic analytical load cases. The examined variations included: Shell and solid mesh, different degrees of discretization, element formulations. All models have been run in LS-DYNA, some selected ones as well in PAMCRASH and ABAQUS Standard, for reasons of comparison. Two basic load cases have been examined:

a) Prismatic cantilever beam with rectangular cross section, under a single force load.
b) Circular plate, fixed at the boundary, under pressure load.

These two models have been chosen in order to have an analytical solution to compare the numerical results with. All models have been run under linear conditions: Elastic, isotropic material behaviour and small deformations. The solution of each variation has been compared to the analytical one.

2. Load cases

The two chosen load cases are a rectangular beam under concentrated load as well as a circular plate under distributed (pressure) load. Both load cases represent typical loading situations in structural analysis. The analyses have been performed in linear conditions, both for material and for geometry. For this assumptions analytical solutions are available.

a. Rectangular beam under bending load

The rectangular beam has been defined with the following dimensions:

Length l = 100 mm, width b = 10 mm, height h = 5 mm.

The boundary conditions are: Fixation in all degrees of freedom on one end, force of F = 20 N on the opposing end.

Youngs Modulus has been chosen as E = 69000 MPa, Poisson's Ratio as μ = 0.3.

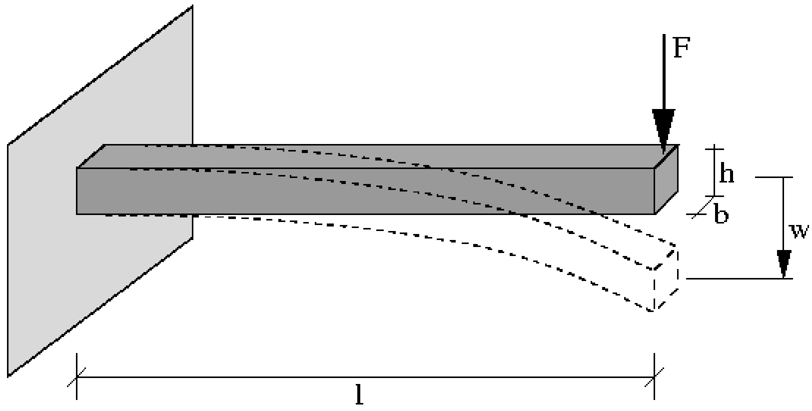

Figure 1: Dimensions of the cantilever beam

The analytical solution is given as:

Deflection: $$w = \frac{Fl^3}{3EI}$$ with *I* being the section

modulus to : $$I = \frac{bh^3}{12}$$.

The maximum stress is defined as: $$\sigma_b = \frac{M_b}{W_b}$$ with

$$M_b = Fl$$

and $$W_b = \frac{I}{h/2}$$

For the given dimensions and stiffness this yields to $w = 0{,}9275mm$ and $\sigma_b = 48MPa$.

b. Circular plate under pressure load

The circular plate has been defined with the following dimensions:

Radius R = 50 mm, height h = 2 mm (see Figure 2).

The boundary conditions are: Fixation in all degrees of freedom at the outer edge of the plate, distributed load of p=0.25 MPa on the whole surface.

Youngs Modulus has been chosen as E = 69000 MPa, Poisson's Ratio as μ = 0.3.

The analytical solution is given by:

Deflection: $$w = 0.171\frac{pR^4}{Eh^3}$$

Stress in center of plate: $$\sigma_r = \sigma_t = 0.488\frac{pR^2}{h^2}$$

For the given dimensions and stiffness this yields to $w = 0{,}484mm$ and $\sigma_b = 76.25MPa$.

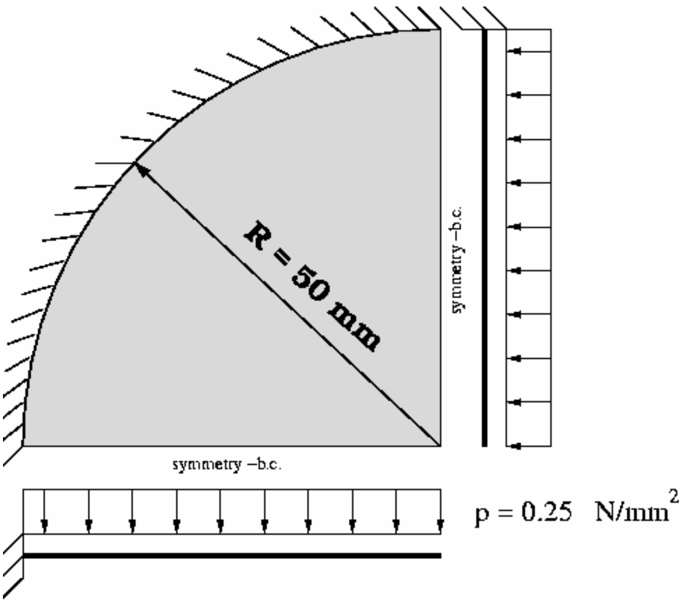

Figure 2: Boundary Conditions of the plate

3. Chosen models and element types

For the two chosen examples three different kinds of discretization levels for all element types are considered, see Figure below. The selected LS-DYNA element types are the most popular and the mainly used ones. For beams type 1 (Hughes-Liu, cross section integration) and type 2 (Belytschko-Schwer, resultant), for shells type 2 (Belytschko-Tsay, one point integrated) and type 16 (fully integrated), for solids type 1 (hexahedron, constant stress - one point integration), type 2 (hexahedron, selective reduced integrated) and type 10 (tetrahedron, one point integrated). For detailed informations to the different element types in LS-DYNA it is referenced to [3] and [4].

For the reduced integrated elements hourglass control is applied. The stiffness option is used for the Belytschko-Tsay shell element and the viscous option is selected for the solid element type 1. The used LS-DYNA version for the explicit caculations is V960-Rev.1106, for the implicit calculations the latest pre-version of V970 is applied. For the PAM-CRASH calculations (V2001) comparable element types with regard to the LS-DYNA element types have been chosen. The tetrahedron element of ABAQUS is a 10 node element (C3D10) with quadratic shape functions.

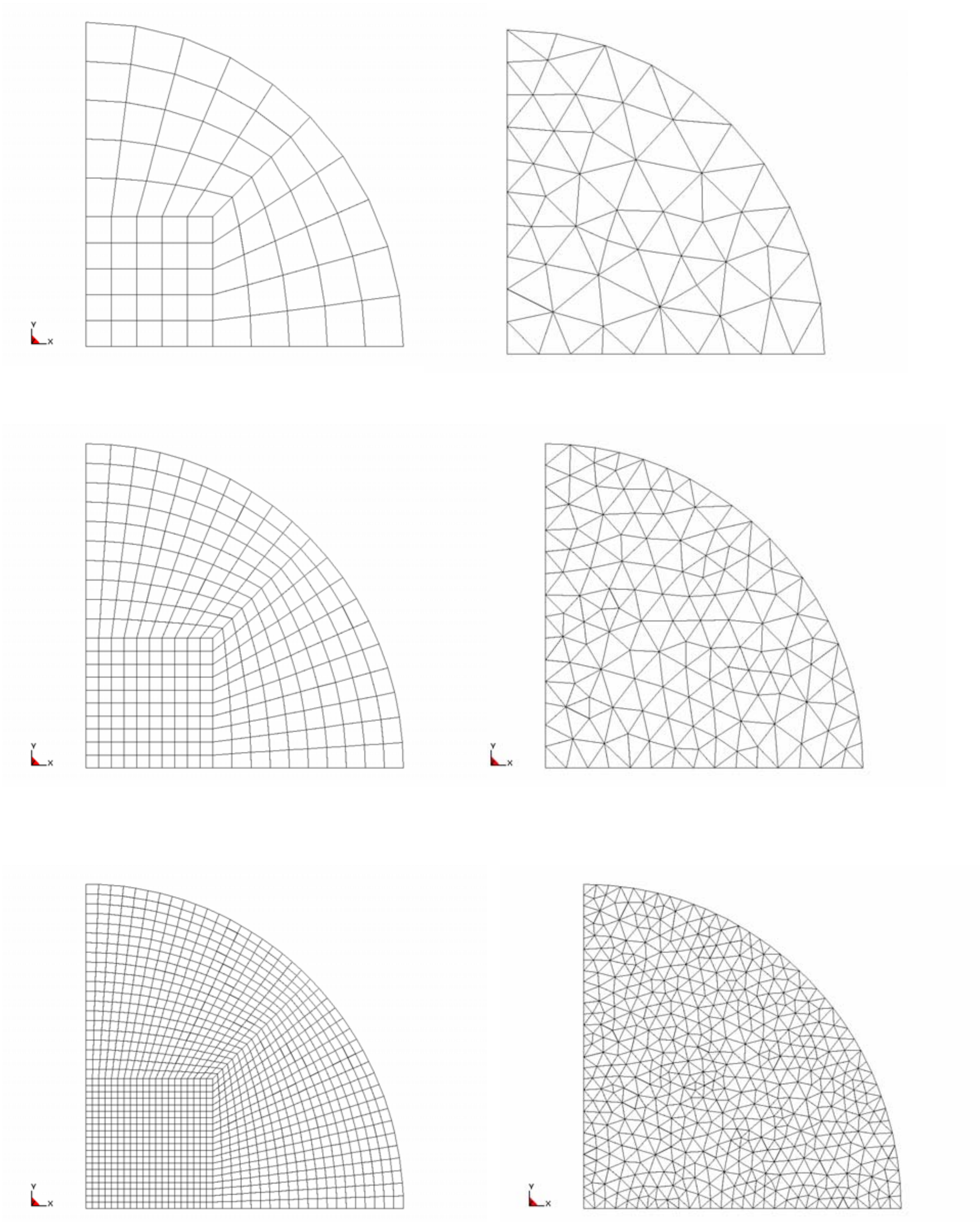

Figure 3: Finite-Element Discretization of one quarter of the plate

4. Numerical Results

The results of both problems the cantilever beam and the circular plate are summarized in a table to be found at the end of this paper.

a. Cantilever Beam

The stress values of the beam in the axial direction are given on the surface at the clamped end. They are calculated by extrapolation of the stress values given at the integration points, see Figure 4. This is only valid for linear material behaviour. For the resultant beam formulation (el.-type 2) the stress is given by the moment at the clamped end. The aspect ratio of the simulations with the quadrilateral shell elements is for the finer discretization very poor. Probably this is the reason, that the results become worse for the finer models.

b. Circular Plate

The stress values of the plate are evaluated at the center of the plate. The values are also extrapolated onto the element surface. The radial and the tangential stress values are equal to the stress values in x- and in z-direction (axisymmetry).

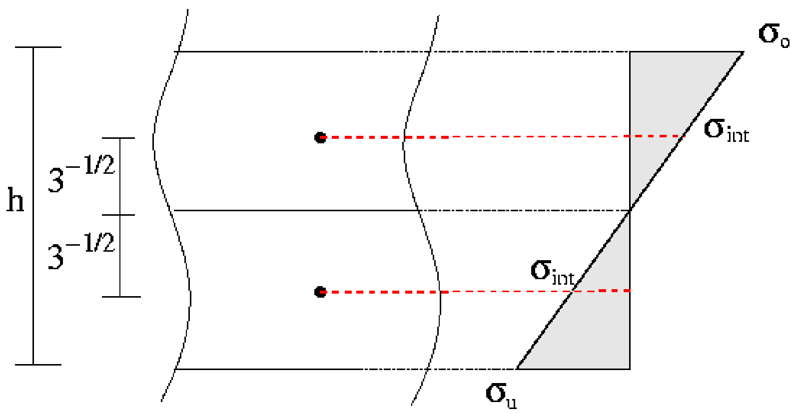

Figure 4: Extrapolation of stress values for a shell element with two Integration Points through thickness

5. Conclusions

The present paper demonstrates, that for a sufficient fine discretization the results of the explicit calculations are fairly close to the analytical solutions. An exception is for the tetrahedron elements of the cantilever beam. This elements behave very poor for the case of bending and shear loading. The 10 nodes quadratic tetrahedron element of ABAQUS overcomes this problem.

The constant stress hexahedron elements of LS-DYNA give for the coarse discretization a much too soft response, but proceed with increasing refinement of the discretization towards the analytical solution. The fully integrated LS-DYNA elements behave in the opposite way, they are too stiff for the coarse discretization and become better with finer discretization.

6. References

[1] Dubbel, Taschenbuch für den Maschinenbau, Chapter C, 1995

[2] LS-DYNA User’s Manual Version 960

[3] LS-DYNA Theory Manual

[4] ABAQUS /Standard User’s Manual, Version 6.1

[5] PAM-CRASH, Version 2001, Solver Reference Manual