Struct Multidisc Optim (2007) 33: 1–12
DOI 10.1007/s00158-006-0040-z

RESEARCH PAPER

Jimmy Forsberg · Larsgunnar Nilsson

Topology optimization in crashworthiness design

Received: 1 February 2005 / Revised manuscript received: 14 February 2006 / Published online: 21 October 2006

Abstract Topology optimization has developed rapidly, primarily with application on linear elastic structures subjected to static loadcases. In its basic form, an approximated optimization problem is formulated using analytical or semi-analytical methods to perform the sensitivity analysis. When an explicit finite element method is used to solve contact–impact problems, the sensitivities cannot easily be found. Hence, the engineer is forced to use numerical derivatives or other approaches. Since each finite element simulation of an impact problem may take days of computing time, the sensitivity-based methods are not a useful approach. Therefore, two alternative formulations for topology optimization are investigated in this work. The fundamental approach is to remove elements or, alternatively, change the element thicknesses based on the internal energy density distribution in the model. There is no automatic shift between the two methods within the existing algorithm. Within this formulation, it is possible to treat nonlinear effects, e.g., contact–impact and plasticity. Since no sensitivities are used, the updated design might be a step in the wrong direction for some finite elements. The load paths within the model will change if elements are removed or the element thicknesses are altered. Therefore, care should be taken with this procedure so that small steps are used, i.e., the change of the model should not be too large between two successive iterations and, therefore, the design parameters should not be altered too much. It is shown in this paper that the proposed method for topology optimization of a nonlinear problem gives similar result as a standard topology optimization procedures for the linear elastic case. Furthermore, the proposed procedures allow for topology optimization of nonlinear problems. The major restriction of the method is that responses in the optimization formulation must be coupled to the thickness updating procedure, e.g., constraint on a nodal displacement, acceleration level that is allowed.

J. Forsberg (✉) · L. Nilsson
Division of Solid Mechanics, University of Technology,
Linköping, Sweden
e-mail: jimmy.forsberg@volvo.com

Keywords Topology optimization · Explicit finite element analysis · Contact–impacts · Nonlinear problems

1 Introduction

Topology optimization is usually meant to be the optimal redistribution of material within a given domain. Actually, changing the topology of a structure is associated with changing its appearance, and unless holes or limbs are created during the optimization, there is no change in the topology. Topology optimization has been the subject of many investigations mainly for static, linear elastic problems (see Bendsøe and Sigmund 2003; Eschenauer and Olhoff (2001) for overviews of the state of the art in topology optimization techniques).

Another approach to topology optimization is to utilize sizing optimization of truss or frame structures. This approach has been used in several investigations, e.g., Sigmund (2000a,b) and Fredricson et al. (2003). Within sizing optimization, nonlinear phenomena such as geometrical nonlinearity have been studied (see Bruns and Sigmund 2004; Buhl et al. 2000), plasticity (see Pedersen 2004), and the formation of plastic hinges for frame structures (see Pedersen 2003). In Pedersen (2004), the crashworthiness property of a frame structure is analyzed using beam elements in an implicit time integration scheme. The gradients of the optimization problem are evaluated using direct analytical differentiation.

This work was inspired by Ebisugi et al. (1998), Soto (2001a,b, 2002). These authors have investigated a method with a parameterized material law to determine the material distribution in a given spatial domain. They have also varied the thickness of shell elements to determine the optimal topology of a structure subjected to impact. The optimal structure for energy absorbing in a vehicle design should fulfill several criteria. The main objective is to absorb energy with a minimum amount of material in a controlled way.

In this paper, a simplified topology optimization method to be used at early design stages is presented. The Internal Energy Density (IED) distribution is investigated and used as a measure on to which extent a certain finite element contributes to the total internal energy and, thus, to the importance of the element from a topological point of view. In the linear elastic case, a related problem is found in minimizing the maximum stress in a structure. The optimal solution of this optimization problem is a structure with an evenly distributed internal energy density. The optimal solution found in the two optimization formulations, i.e., minimize compliance and minimizing the maximum stress, is not identical.

The methods proposed here have been used to find an optimal distribution of material in a structure subjected to multiple loadcases.

In the following, we have studied the topology optimization of 2-D plane stress problems. The methods can, however, be generalized to 3-D.

Although a linear elastic structure never can be used as an energy-absorbing device, we have included some linear elastic cases to evaluate the implemented methods and to compare to existing methods for this class of problems. In Section 4, many aspects of the topology optimization procedure are developed and analyzed for a static loadcase, which is solved both using explicit time integration and a static solver. In Section 5, a dynamic loadcase is studied and the dynamic loads are in for the static solver interpreted as pressure loads.

2 Theory of topology optimization in the linear elastic case

The standard formulation of a linear elastic, finite element (FE) discretized, topology optimization problem is to minimize the external work, which in the linear elastic case corresponds to the minimum strain energy, i.e.

$$(P)\quad \begin{cases} \min\ \mathbf{F}^T\mathbf{u} \\ s.t.\ \begin{cases} \mathbf{K}(\rho)\mathbf{u} = \mathbf{F} \\ \rho^T\mathbf{A} = V \\ \rho_{min} \le \rho_e \le \rho_{max},\ \ e = 1, \ldots, n_{el} \end{cases} \end{cases} \tag{1}$$

where $\mathbf{F}$ is the vector of external nodal forces acting on the structure, $\mathbf{u}$ is the vector of nodal displacements, $\mathbf{K}$ is the stiffness matrix, $\boldsymbol{\rho} = \mathbf{t}/t_{max}$ denotes the scaled thickness of each element or rather the relative density (the design variables), $\mathbf{A}$ is the area of each element and V is the total volume of material, which is available to construct the structure.

With this formulation, an optimal thickness distribution may be found. However, the thickness distribution may be diffuse in certain regions. To get a more distinct material distribution indicating material or no material, the intermediate values of thicknesses can be penalized. This penalization will make the optimization problem non-unique, in the sense that the optimal solution asymptotically moves toward a value; however, unless the solution space is restricted somehow, the optimal point will never be found. In this paper the traditional SIMP penalization method was used. There are also other mathematical problems associated with this optimization technique, e.g., *checkerboard patterns* and *mesh dependency*. These issues are discussed in, e.g., Sigmund and Petersson (1998), Petersson (1998), Borrvall and Petersson (2001), and Bendsøe and Sigmund (2003).

3 Proposed methodology for topology optimization in crashworthiness design

The topology optimization method presented here is based on the Internal Energy Density in a finite element. In a topology optimization formulation, one might use the objective to minimize the maximum stress, $min(max(\sigma))$, of any element in the structure at hand. This formulation is related to finding a structure with a uniform IED intensity. Therefore, the IED in each element can be used as a measure of how much it contributes to the total internal energy of the structure.

The approach in this paper is to use the IED parameter to determine whether an element is efficient or not from an energy point of view. If it is inefficient, the element is either completely removed or its thickness is reduced. On the contrary, if the element is efficient it is either kept to the next iteration of the optimization process or the thickness of the element is increased.

The basic assumption of this method is that the stress state surrounding a finite element remains the same in two consecutive iterations to motivate the thickness update. This is hardly true for all possible loading situations and all possible optimization histories.

The optimization formulation for the methodology with deletion of finite elements can be stated as

$$(P)\quad \begin{cases} \max(\min(IED^{el})) \\ s.t.\ \begin{cases} M\ddot{u} + f_{int} = f_{ext} \\ IED \le IED_{max} \\ n_{el} \ge n_{min} \end{cases} \end{cases} \tag{2}$$

where IED^{el} is the internal energy density in an element found in the structure. M is the mass matrix and $\ddot{u}$ are the nodal accelerations, f_{int} and f_{ext} are the internal and external nodal load vectors, respectively. n_{el} is the number of elements used in the finite element model and n_{min} is the minimum number of elements allowed for the solution for apparent reasons. The internal energy density is defined as

$$IED = \int_0^t \boldsymbol{\sigma} : \mathbf{D}dt \tag{3}$$

where $\boldsymbol{\sigma}$ and $\mathbf{D}$ are the Cauchy stress and rate-of-deformation tensors, respectively, and t denotes time. The IED will increase with the deformation as long as the material undergoes hardening and no elastic unloading takes place.

Similarly, the optimization problem for the methodology with variable finite element thicknesses is stated as

$$(P)\quad \begin{cases} \min(\sum_{i=1}^{n_{el}}(IED_i - IED_{target})^2 \\ s.t.\ \begin{cases} M\ddot{u} + f_{int} = f_{ext} \\ \rho_{min} \le \rho_i \le \rho_{max},\ \ i = 1, \ldots, n_{el} \end{cases} \end{cases} \tag{4}$$

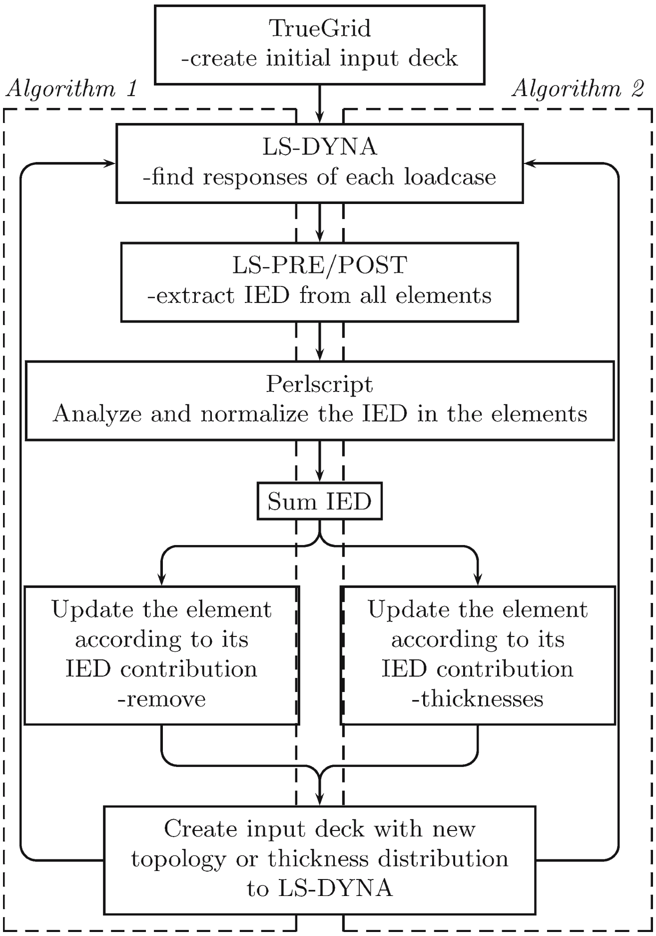


Fig. 1 Scheme of proposed topology optimization procedures. Observe there are two algorithms in the pigure with many similar steps

where IED_{target} is a user-set target value for all elements. A volume or mass constraint for the structure can be added in the optimization problem for a more general formulation.

3.1 Evolution of the topology

The proposed topology optimization approaches do not explicitly use gradient information to solve the problem. Instead, in each iteration, the IED value in each element is used to determine whether this element is efficient or not for the loading cases at hand. The iterative procedures are shown in Fig. 1.

The initial model (ground structure) is created using TrueGrid (see Rainsberger 2001) and the impact analysis is solved using the explicit FE solver LS-DYNA (see Hallquist 1998, 2004). The post-processing is done using LS-PRE/POST (see Hallquist 2002) and different Perl scripts. Perl scripts are also used to update the FE model.

A stop criterion of the scheme in Fig. 1 must be given. Any criterion suitable for the purpose at hand can be used, e.g., the maximum number of removed elements, a minimum change in relative thickness, a minimum change in the IED distribution, etc.

3.2 Multiple loadcases

In the case of multiple loadcases, our approach is to normalize all element IEDs with the maximum IED value found in any element for that particular loadcase. In this work, the IEDs are normalized for each loadcase and the IEDs from each loadcase are then summed for each element.

$$IED(j) = \sum_{LC} \alpha_{LC} IED_{LC}(j) \tag{5}$$

where $IED(j)$ is the summed IED for element j, LC denotes the number of loadcases, and α_{LC} is a loadcase weighting factor (usually $\alpha_{LC} = 1$). The summed IED is then evaluated and the elements are modified/removed according to its summed IED (see Fig. 1). This will give a small load more influence on the final structure than is motivated through the energy density generated by it. Ideally, every load has a set of criteria to fulfill and the value of these criteria should be accounted for. However, in this application, every loadcase is given equal weight.

3.3 The FE model updating procedure

An algorithm controls the update of the topology. In the case of removing elements, a percentage of the maximum IED found in any finite element is used as an indicator to remove elements with lower IEDs. This percentage has to be given by the user.

In the case of thickness update, another approach is used. The basic idea is to set a target value for the IED. If an element has a higher IED summed over all loadcases, the thickness is increased; if it has a lower IED, it is decreased. A range, defining the maximum element thickness change within an iteration, is used. Also a factor is defined for each element

$$q = \frac{IED^i}{IED_{max}} \tag{6}$$

where IED^i is the summed internal energy density of the currently updated element and IED_{max} is the maximum summed internal energy density found in any element

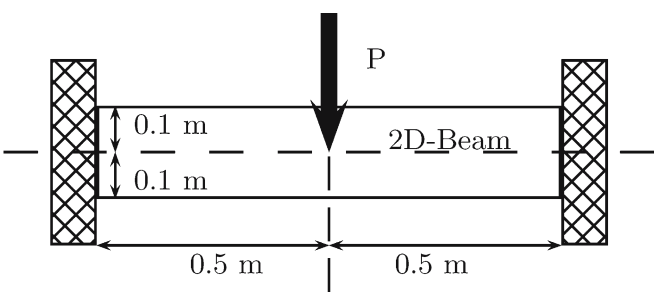


Fig. 2 2-D plane stress structure subjected to a point load P

Table 1 Material parameters used for the 2-D plane stress structure

Parameters	
E	210 GPa
ν	0.3
σ_Y	500 MPa
$Etan$	1,000 MPa
ρ	7,800 kg/m^3

in the structure. Depending if q is greater or lower than IED_{target}/IED_{max}, a factor f is defined as

$$f = \begin{cases} \frac{IED^i - IED_{target}}{IED_{target}}, q \leq IED_{target}/IED_{max} \\ \frac{IED^i - IED_{target}}{IED_{max} - IED_{target}}, q \geq IED_{target}/IED_{max} \end{cases} \quad (7)$$

and the new element thickness is set according to

$$t_{new} = t_{old} + f * range \quad (8)$$

where range is a user-defined parameter. Finally, if the global limits of the element thickness are violated, the thickness is reset to the limit value. This kind of updating technique can be seen as a traditional panning scheme, where the global limits of the thickness define the design domain and the range defines a region of interest.

Of course, the target value of the IED will determine in which direction the optimization procedure should go. The range parameter will affect the convergence rate of the optimization problem.

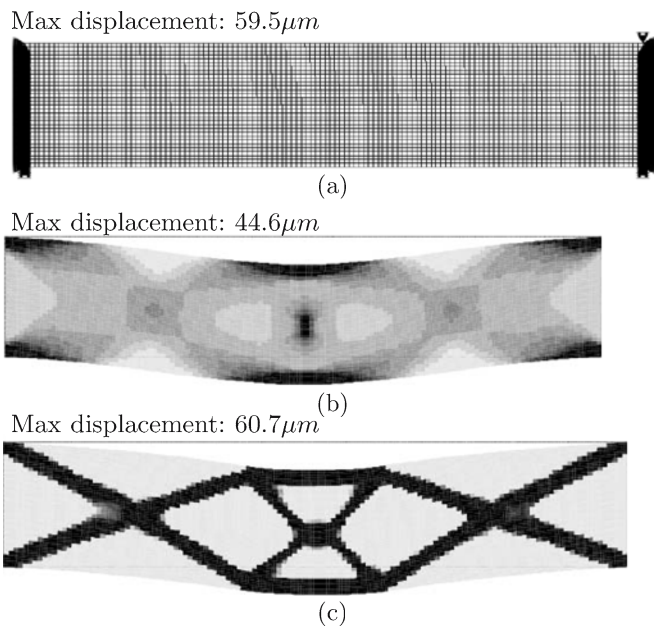


Fig. 3 The 2-D plane stress FE model. **a** Ground structure, **b** optimization results without penalization of intermediate thickness values, **c** optimized result with penalization of intermediate thickness values. *Black* indicates areas with a high thickness

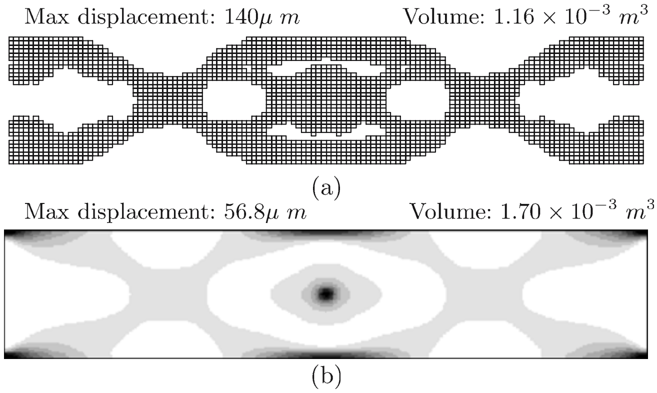


Fig. 4 The optimized 2-D plane stress structure obtained with the two methodologies. **a** Deletion of elements, **b** thickness optimization. *Black* indicates $\rho \geq 0.6$ and *white* indicates a $\rho \leq 0.01$

4 Application problem I: the 2-D plane stress problem

The ground structure is a 2-D plane stress structure subjected to a point load at its midpoint and it has two clamped boundaries (see Fig. 2). The initial volume of the structure is 2×10^{-3} m^3. The magnitude of the point load is initially set

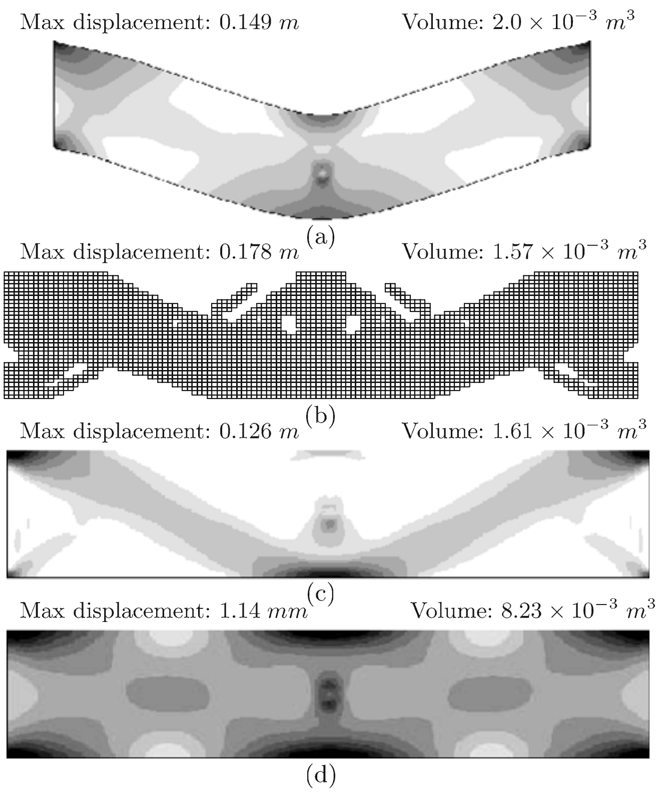


Fig. 5 **a** The distribution of effective plastic strain in the initial model. *Black* indicates a value of at least 40%, **b** the resulting topology using element deletion, **c** the thickness distribution obtained with a high IED target value (the average value of the IED in the initial structure with the high load intensity). *Black* indicates a $\rho \geq 0.5$ and *white* indicates a $\rho \leq 0.01$. **d** The thickness distribution obtained with a low IED target value. *Black* indicates a $\rho \geq 0.9$ and *white* a $\rho \leq 0.03$

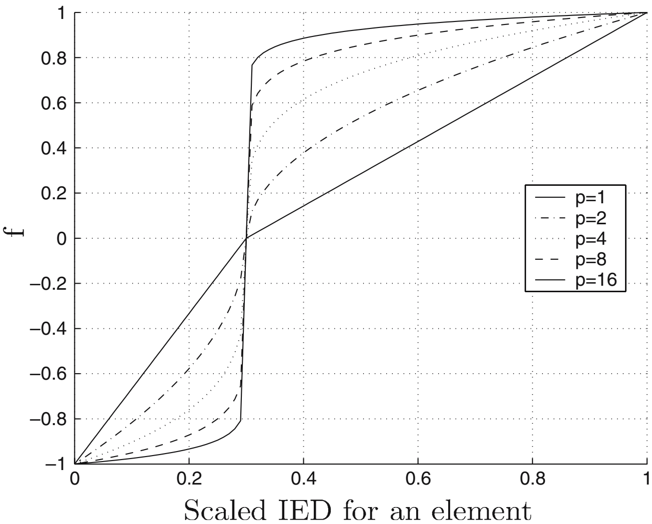


Fig. 6 The influence of the penalty parameter on the factor used in the updating scheme of the thicknesses

to 10 kN, but it will subsequently be increased to introduce plastic material behavior.

The structure is made of steel and the material parameters are given in Table 1.

This problem is solved using both the static, linear elastic FE solver Trinitas (see Torstenfeldt 1998) and the nonlinear FE solver LS-DYNA (see Hallquist 2004). In the 2-D linear elastic case, this is a rather standard topology optimization problem defined by (1). This application problem gives us a possibility to compare the proposed solution technique to the generally used gradient-based method. In addition, we demonstrate the effect of plastic material behavior on the topology evolution.

4.1 Topology optimization in the static, linear elastic case

The structure is subjected to a point load, $P = 10$ kN, at its midpoint, and four noded bilinear elements are used for the mesh. The ground structure and the topology optimization

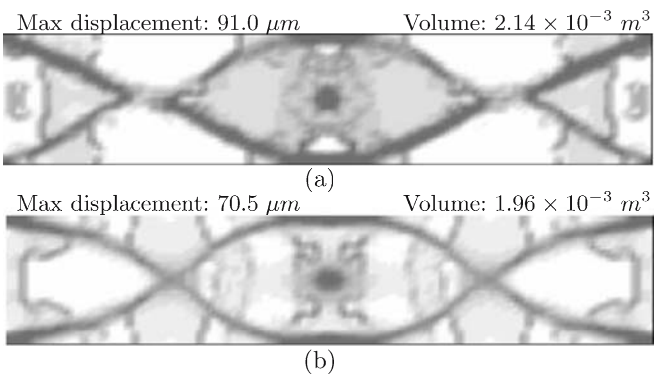


Fig. 7 The optimal structure obtained using the penalty formulation. *Black* indicates $\rho = 1$ and *white* indicates $\rho = 0.003$. **a** Penalty $p = 8$, **b** Penalty $p = 4$

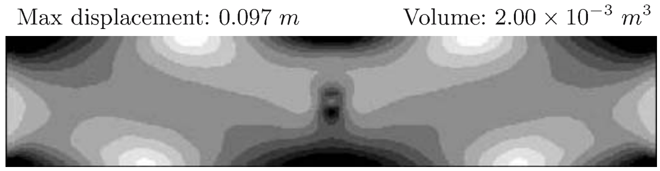


Fig. 8 The optimal structure of the 2-D plane stress structure subjected to a high load intensity and a volume constraint. *Black* indicates $\rho \geq 0.67$ and *white* indicates $\rho \leq 0.03$

results are given in Fig. 3. The thickness of the structure is initially set to 10 mm.

There exists some regions in this structure that are likely to deform plastically, if the loading is sufficiently high or if the element thicknesses in these regions are too small. These regions are located at the fixed boundary corner points and at the point of loading. Considered as a beam, the bending moment at two points along its length tends to zero. These are also two locations where it is likely that hinges are formed if the structure can deform plastically.

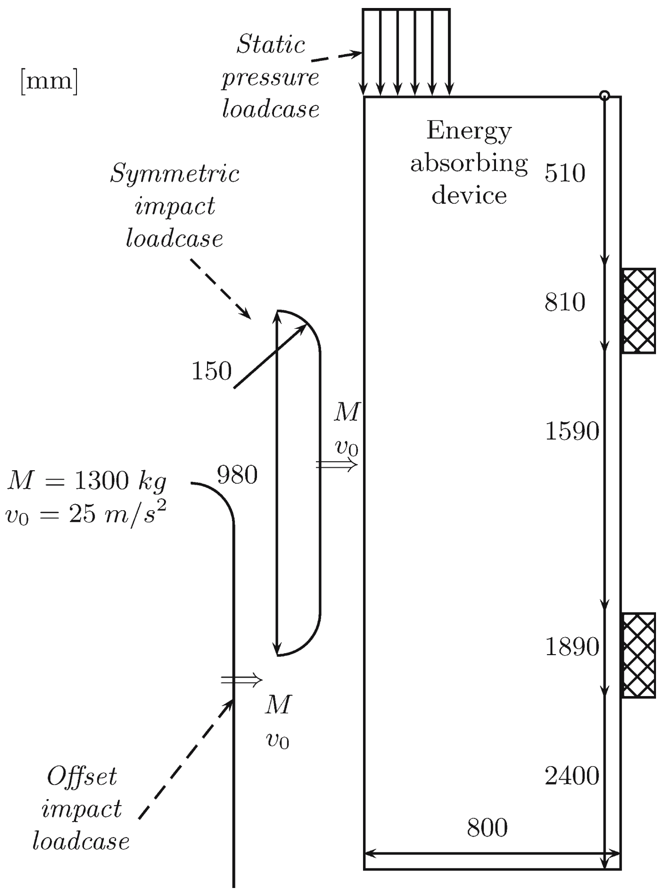


Fig. 9 The ground structure of the energy absorbing device subjected to three loadcases

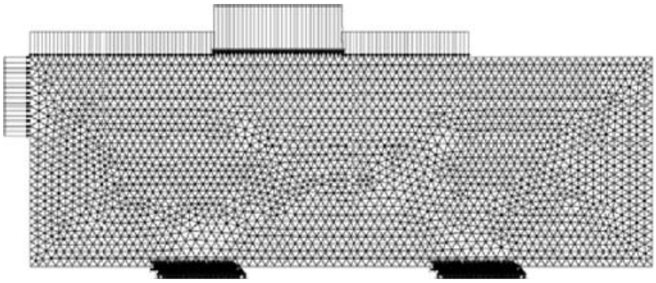


Fig. 10 Finite element model of the ground structure

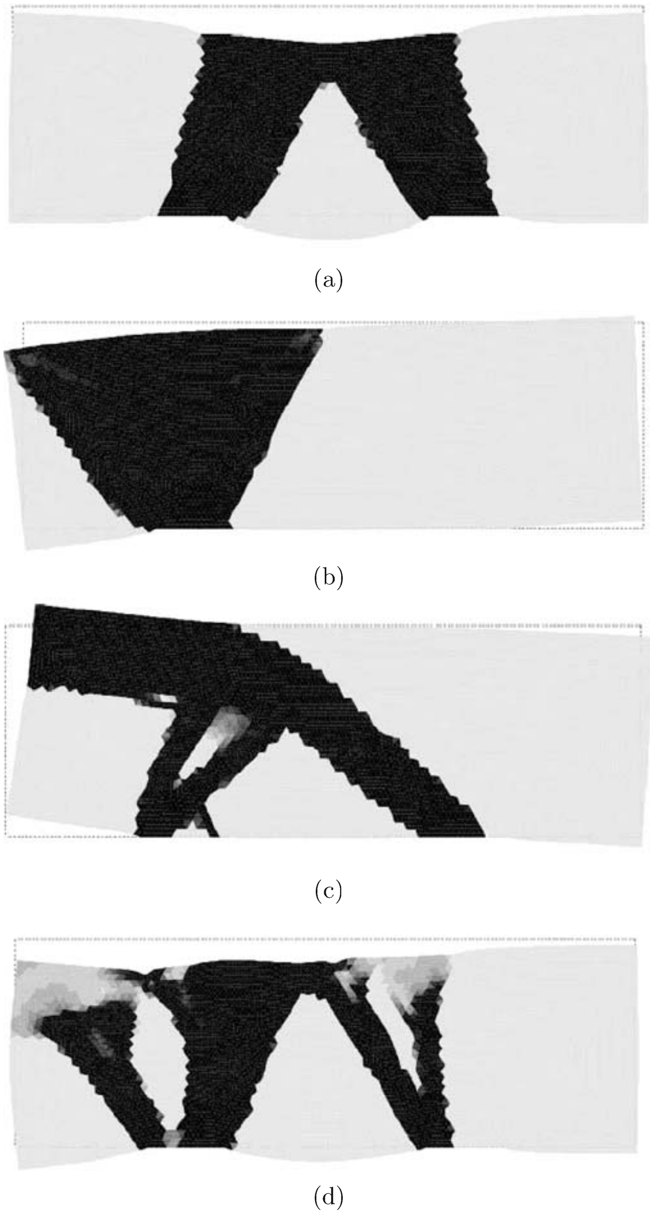


Fig. 11 Results from linear elastic topology optimization. *Black* indicates thick elements and *white* indicates no material. **a** The static symmetric loadcase, **b** the static offset loadcase, **c** the pressure loadcase. **d** All three loadcases simultaneously

4.2 Topology optimization in the nonlinear case

In this section, we utilize the techniques described in Section 3 to optimize the topology of the 2-D plane stress structure (see Fig. 2). The simulations are carried out with an explicit solver and the load is ramped up to its final value. Four noded, bilinear, plane stress elements are used. First, two topology optimization processes are done with a low loading intensity such that the material behaves elastically linear. Next, the structure is subjected to a higher load intensity (distributed on ten nodes around the center of gravity of the structure). This introduces a high level of plasticity into the material. The topology obtained with the low load intensity is used as an initial model in a topology optimization using the thickness optimization formulation. In the different optimizations processes where the element thickness is altered, the minimum thickness is set to 1 mm and the maximum thickness is set to 100 mm.

4.2.1 2-D plane stress structure optimized with a low load intensity

The 2-D plane stress structure was optimized using the two methods presented in Section 3. The obtained topologies are shown in Fig. 4. The results clearly indicate an optimal structure similar to the one obtained when minimizing the external work in the linear elastic case. For the thickness optimization, the target value of the IED was set to 165 Nm/m^3, which is the average value of the IED for the elements in the structure in the initial simulation.

4.2.2 2-D plane stress structure optimized with a high load intensity

In these optimizations, the load intensity is increased such that large plastic strains develop. By deleting elements, two main regions, which are mainly loaded in tension, are obtained. A similar topology is obtained when using the thickness optimization technique if a high value of the IED target value is set. If the IED target value is set to the same value as in the low load intensity case, 165 Nm/m^3, the resulting structure is similar to the one found by compliance minimization, even though the initial structure had high plastic strains (see Fig. 5). The opposite is also true, if the optimization starts with the optimal results found with the low load intensity but now using the high load intensity, the optimal topology shown in Fig. 5c is found.

4.3 Penalization of intermediate thickness values

From a manufacturing point of view, it is difficult and expensive to construct a part with too much thickness variation. Therefore, a thickness penalization formulation was investigated to get a more distinct topology.

The factor determined in (7) can be viewed as a linear penalization depending on the IED value. Our thickness

Table 2 Material properties used for the energy-absorbing device

Properties	
E	100 GPa
ν	0.3
σ_Y	50 MPa
$Etan$	200 MPa
ρ	4,800 kg/m^3

penalization approach modifies the factor f in (7). Each element thickness is updated using a factor

$$f_{new} = (f_{old})^{1/p}, \tag{9}$$

where $p \geq 1$ is the penalization parameter. If p is set to 1, the linear factor is retained. If $p > 1$ then most of the intermediate thicknesses are set toward the limits of the range. In Fig. 6, the factor f is plotted against IED for different values of the penalization factor.

4.3.1 Results from the penalization study

The 2-D plane stress structure with a low load intensity is considered. The global limits of the thickness are lowered, i.e., the minimum allowed thickness is 0.1 mm and the maximum is 30 mm. We are using a range of 5 mm. With the penalization value of 4 and 8, the topologies shown in Fig. 7 are obtained after 26 iterations.

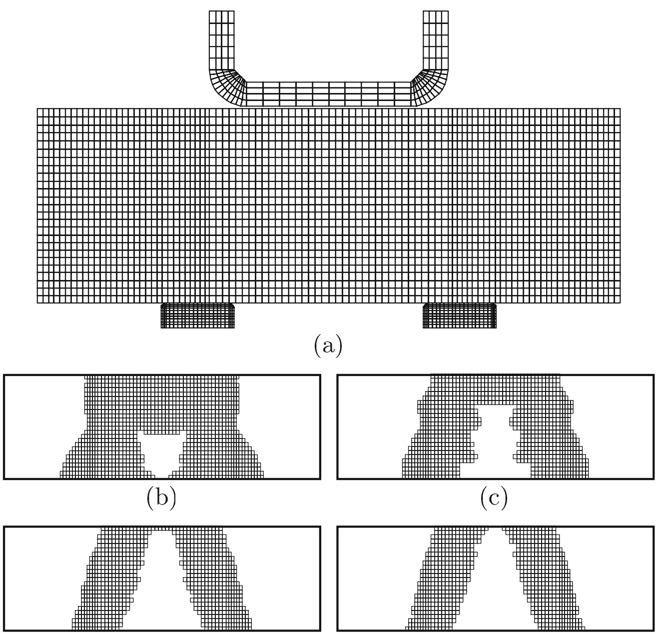


Fig. 12 The topology of the structure during the optimization process for the symmetric frontal impact. **a** The initial configuration, **b** second iteration, **c** third iteration, **d** fifth iteration, **e** sixth iteration

Table 3 The lower limit of IED and elements removed in each iteration of the symmetric frontal impact

Iteration	$\frac{lower\ limit}{max\ IED}$	Numbers of elements removed
1	0.5	1,316
2	0.5	200
3	1	88
4	5	78
5	5	44

4.4 Volume constraint in the nonlinear case

In the previous optimization formulations, there was no constraint on the available amount of material. Hence, the total mass could increase as in the 2-D plane stress thickness optimization problem. In this section the total amount of volume is kept below a certain level. The finite element thicknesses are updated as before. To restrict the volume material used, however, every finite element thickness is scaled with the factor (if the volume constraint is broken)

$$s = 1 - \frac{V - V_{all}}{V} \tag{10}$$

where V is the volume after the thicknesses have been updated with respect to the IED distribution and V_{all} is the total amount of available volume. Since some of the elements might be on the lower global limit of the thickness value, these thicknesses are reset to the limit value. Therefore, the volume limit will still be broken. To minimize this constraint violation, the new volume is calculated and a new factor is found. Hence, we can find iteratively the thickness distribution, which also fulfills the volume constraint.

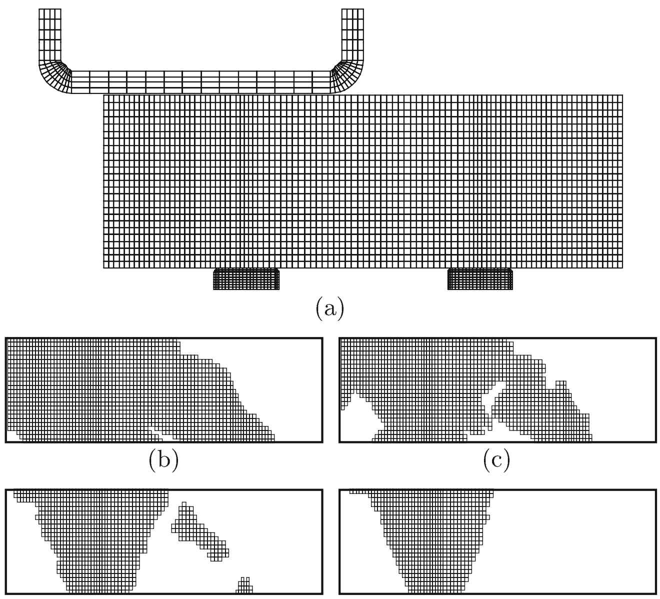


Fig. 13 The topology of the structure during the optimization process for the offset frontal impact. **a** The initial configuration, **b** second iteration, **c** third iteration, **d** fifth iteration, **e** seventh iteration

Table 4 The lower limit of IED and elements removed in each iteration of the offset frontal loadcase

Iteration	$\frac{lower\ limit}{max\ IED}$	Numbers of elements removed
1	0.001	1,646
2	0.01	235
3	0.1	268
4	0.5	281
5	1.0	162
6	1.0	23

Table 5 The lower limit of IED and elements removed in each iteration of the pressure loadcase

Iteration	$\frac{lower\ limit}{max\ IED}$	Numbers of elements removed
1	0.001	401
2	0.005	431
3	0.01	164
4	0.02	247
5	0.03	168

4.4.1 Application with an active volume constraint

The 2-D plane stress structure (see Fig. 2) with a high load intensity and a low value of the target IED was used earlier. This formulation of the topology optimization problem generated a primarily elastic solution although the finite element thicknesses were increased. Hence, a heavier structure was found (see Fig. 5). The same ground structure and topology optimization formulation are used but with a volume constraint, and the allowed volume is the volume of the ground structure. The optimized structure is shown in Fig. 8. An issue concerning the volume constraint is that all element thicknesses are initially scaled and the lower limit is then checked. The elements having a thickness equal to the upper limit at that time could be excluded in the scaling procedure, since the results have been indicating that these elements are vital. Likewise, the element with a thickness equal to the lower limit should be excluded since this thickness will be set to the lower limit in the next step.

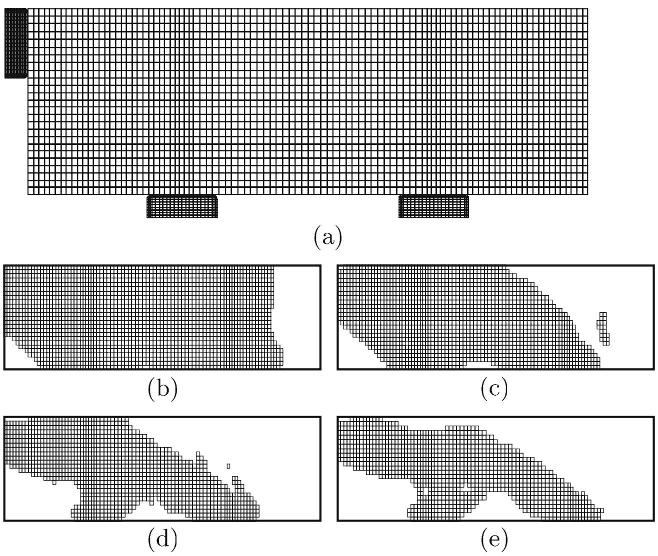


Fig. 14 Successive topologies of the structure during the optimization process for the pressure loadcase. **a** Ground structure and initial configuration, **b** second iteration, **c** third iteration, **d** fifth iteration, **e** sixth iteration

5 Application problem II: energy absorbing device

The task is to develop an energy absorbing frontal underrun protection device for a truck. The structure can occupy a well-defined region in space and it is fixed to the front of the truck. The ground structure and other data are given in Fig. 9. The structure is subjected to three loadcases: two dynamic loadcases, i.e., symmetric frontal impact and offset frontal impact, respectively. In addition, it is subjected to one static transversal uniform pressure loadcase.

5.1 Topology optimization in the linear elastic case

Since an elastic structure cannot absorb energy, the main objective with the topology optimization assuming static loading and linear elastic behavior is to compare these results to results obtained from the topology optimization methodologies applied with impact loading and nonlinear structure.

In this section, all loadcases were interpreted as pressure loadings and six noded triangular elements are used. The topology optimization problem is solved by minimizing the compliance assuming linear elastic material. Six noded triangular elements are used. A penalty formulation is used to penalize the thicknesses between zero and the maximum value for each element, i.e., either material or no material in different areas of the structure is obtained.

5.1.1 Results from the linear elastic case

The ground structures mesh is shown in Fig. 10. The results from the topology optimization subjected to the three

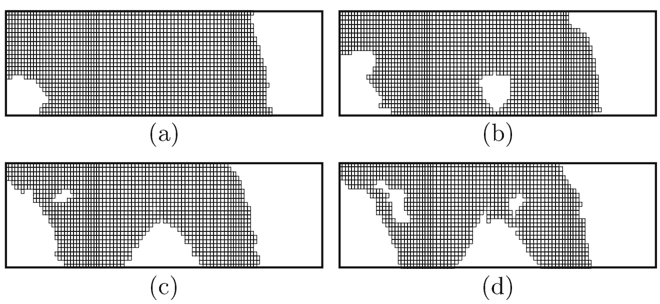


Fig. 15 The topology of the structure during the optimization process for the multiloadcase optimization. **a** Second iteration, **b** third iteration, **c** fifth iteration, **d** sixth iteration

Table 6 The lower limit of IED and elements removed in each iteration of the multiload optimization

Iteration	$\frac{lower\ limit}{max\ IED}$	Numbers of elements removed
1	0.01	597
2	0.1	177
3	0.5	169
4	0.75	106
5	2.0	125

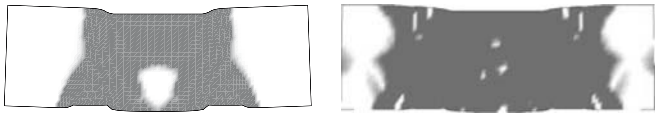
different loadcases separately and the result from a topology optimization subjected to all three loadcases simultaneously are shown in Fig. 11.

5.2 Topology optimization in the nonlinear case—deletion of elements

The ground structure consists of a uniformly distributed fictitious material. The main reason of using a fictitious material is to ensure that reasonable plastic deformations do occur in order to absorb the applied energy. The material is selected to behave like an elastoplastic material (see Table 2 for the material parameter values used).

The structure is subjected to symmetric and offset loadings and a static pressure as shown in Fig. 9. The main scheme of this optimization procedure follows the scheme shown in Fig. 1. To determine if an element should be removed or kept, a percentage of the maximum IED found among the finite elements is used as a lower IED limit. If an element has a lower value of the IED, the element is removed in the next iteration of the optimization process.

The lower limit varies depending on the loadcase and on the stage of the optimization process. If the lower limit is too high, many elements may be deleted and elements that constitute important load paths may also be deleted. Since, there is no way to determine a maximum of the lower limit, its value should be kept low. One idea to determine an appropriate value might be to investigate and limit the number of elements removed in each iteration.



(a) (b)

Fig. 17 IED distribution in the structure for the initial (**a**) and final state (**b**) for the symmetric frontal impact. *Black* indicates that $IED \geq 100\ kNm/m^3$

5.2.1 Results symmetric impact—deletion of elements

The ground structure and initial configuration of the problem are given in Fig. 12. The topologies of the energy-absorbing device at selected iteration steps are also shown in Fig. 12. The similarity with the result obtained in the linear elastic case is obvious. The lower limits of IED are kept low in order not to remove too many elements in one iteration.

In Table 3, the percentage of maximum IED found during the iterations and elements removed in each iteration are given.

5.2.2 Results offset impact—deletion of elements

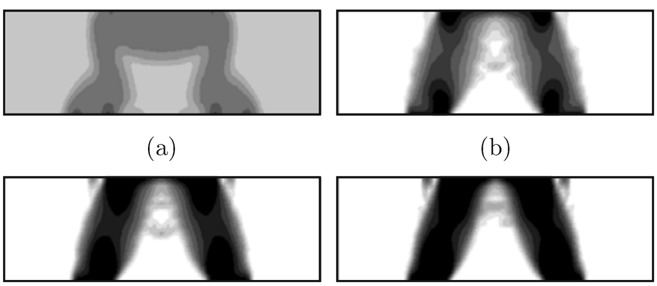
The topologies obtained for this loadcase is similar to that obtained by minimizing the compliance in the linear elastic case. When comparing the topology in Fig. 11 with the topology in Fig. 13, we note that they are very close to each other.

In Table 4, the lower limit of IED and elements removed in each iteration are given.

5.2.3 Results pressure loadcase—deletion of elements

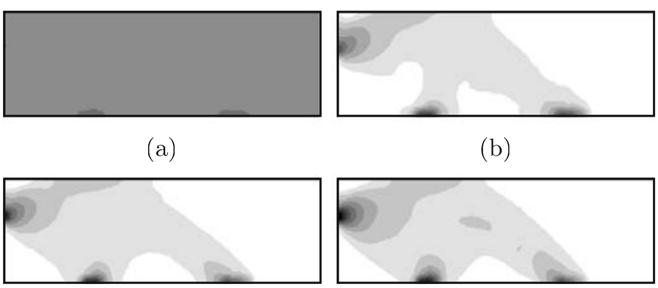
Due to the nature of this loadcase, the IED is high at different discrete load points, i.e., at least relative to other parts of the structure. Therefore, it is in this case necessary to use a low lower limit of the IED such that just a few elements are deleted in the first iterations. Note that the deformations of the initial structure in this loadcase primarily were elastic; hence, it is the opposite to the previous two loadcases.

Comparing the topology obtained from the minimization of compliance with the topology obtained from the nonlinear



(a) (b) (c) (d)

Fig. 16 Thickness distribution in the structure during the optimization process of the symmetric frontal impact. **a** Second iteration, **b** eighth iteration, **c** 16th iteration, **d**: 26th iteration



(a) (b) (c) (d)

Fig. 18 Thickness distribution in the structure during the optimization process for the pressure loadcase. **a** Second iteration, **b** tenth iteration, **c** 18th iteration, **d** 26th iteration

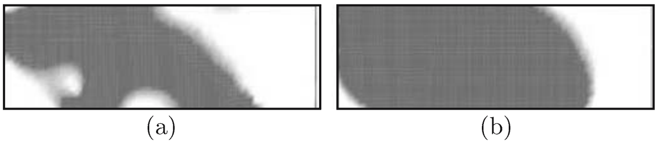


Fig. 19 IED distribution in the structure for the initial (**a**) and final state (**b**) for the pressure loadcase. *Black* indicates that $IED \geq 1\ kNm/m^3$

structural problem, similar structures are obtained, although not as similar as in the impact loadcases, see Fig. 14.

In Table 5, the lower limit of IED used and elements removed in each iteration are given.

5.2.4 Results multiloads—deletion of elements

In this section, the three previous loadcases are combined in a topology optimization, see Fig. 15. The results are similar to the one obtained when minimizing the compliance in the linear elastic case. However, in the nonlinear case the final structure seems to have holes at other locations than in the linear elastic case. In Table 6, the lower limit of IED used and elements removed in each iteration are given.

5.2.5 Conclusions from the topology optimization—deletion of elements

The topologies obtained by minimizing the compliance assuming a linear elastic behavior found to be similar to the topologies obtained by the nonlinear methodology. Thus, if the internal energy density for every element of the structure is maximized (and at the same level), the compliance of the structure will be minimized. This is also true for a nonlinear system.

In the case of the energy-absorbing frontal underrun protection device studied here, the dynamic behavior is shown as less important than the final deformation.

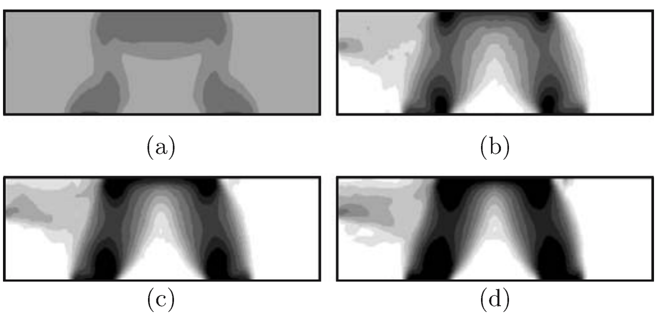


Fig. 20 The thickness distribution in the structure during the optimization process for the multiloadcase optimization. The static pressure loadcase and symmetric frontal impact case are used. **a** Second iteration, **b** tenth iteration, **c** 18th iteration, **d** 26th iteration

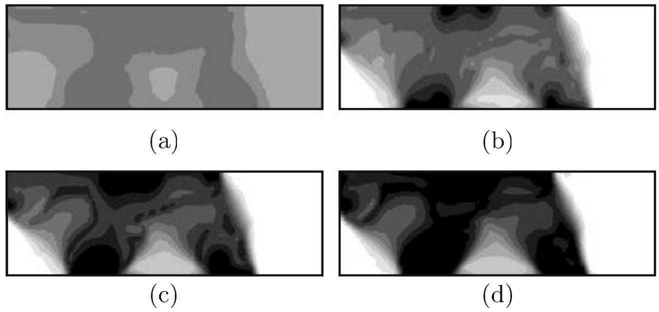


Fig. 21 The thickness distribution in the structure during the optimization process for the multiloadcase optimization. All three loadcases are used. **a** Second iteration, **b** tenth iteration, **c** 18th iteration, **d** 26th iteration

5.3 Topology optimization in the nonlinear case—modifying thickness of elements

The main advantage of modifying the thickness of an element instead of deleting it is that an element with a small thickness is kept in the model and its thickness can grow at a later stage of the optimization process. If an element is deleted, as in the previous formulation, that load path is removed and cannot be reintroduced.

The objective in the present optimization methodology is to find a structure with an evenly distributed IED. A target value for this IED level has to be selected. In the optimization processes presented here, we have used the average value found in the initial FE model. The thickness may be altered within the interval of 1 to 200 mm.

5.3.1 Results symmetric loadcase—modifying thickness of elements

The thickness distributions at four iterations of the topology optimization process are shown in Fig. 16. The maximum allowed change in thickness within an iteration is set to 50 mm. After some initial oscillations, the major topology is revealed. In the following iterations, the thickness distribution moves toward a similar structure as obtained with the element deletion method. The IED target level is set to 2.12 MNm/m^3, which is the average value of the IED for the initial simulation.

The distribution of altered IED is shown in Fig. 17.

5.3.2 Results pressure loadcase—modifying thickness of elements

The thickness distribution at four stages of the thickness optimization of the pressure loadcase are shown in Fig. 18. A maximum change of 10 mm in thickness of an element was allowed. The IED target value is set to 42.8 kNm/m^3, which is the average value of the IED for the initial simulation.

The IED distribution is shown in Fig. 19.

5.3.3 Topology optimization with combined loadcases—modifying thickness of elements

In these optimizations, firstly, the two loadcases investigated earlier with the thickness topology optimization process were evaluated in a combined optimization process according to Section 3.2 see Fig. 20. Secondly, all three loadcases are used in a combined optimization process, see Fig. 21.

The maximum thickness change in one iteration is set to 30 mm throughout the optimization process. The IED target value is set to the sum of the average IED for the individual loadcases.

5.3.4 Conclusions of the procedure with modifying the thickness of elements

The results found with the thickness topology formulation are similar to the results found with the method using deletion of elements. By keeping all elements in the FE model, load paths not important in the early optimization process can reappear. Therefore, the thickness topology optimization process has more capabilities than the method with deleting elements (cf. Bendsøe and Sigmund 2003).

6 Conclusions

We have presented two topology optimization methods that can be applied to nonlinear structures subjected to dynamic loading. Several different loadcases can be combined in the optimization process even if they have different characteristics.

Concerning the optimization method based on deletion of finite elements, it gives an indication of where to put material. Since finite elements are removed, the design space is made smaller as the optimization process proceeds, and we have not allowed the reintroduction of finite elements. Another important issue is the number of elements that can be deleted in one iteration. If too many elements are deleted, important load paths may disappear and, therefore, the number of elements deleted within one iteration should be limited.

To avoid a decreasing design space, an optimization procedure based on the finite elements thicknesses was introduced. Using this approach, we have shown that it converges toward the minimum compliance solution in the linear elastic case. In the impact loading application, it converges toward a similar topology as found using the method with deleting elements. A penalty formulation was introduced, with the intent of achieving more distinct topology results and it is shown to work properly.

With the thickness update methodology, there are some steering parameters to be set and some general observations can be made. The range parameter highly influences the convergence rate of the procedure. If it is set too low, the topology optimization process converges rather slowly but in a linear way. If it is set too high, oscillations in the element thicknesses are observed during the iterations, which also decrease the convergence rate of the optimization process.

With the target value of the IED, the optimization process pushes the solution toward a predefined load intensity in each finite element. A proper value for the IED parameter can be determined from the material properties.

We have observed that the penalty parameter seems to work better if the range parameter (see (8) defines a larger part of the global design domain.

There are issues that were not fully addressed. The influence of the range, target value of the IED, the penalty factor, and the global limits on the thickness are all factors that need further investigations to make the optimization procedures more efficient.

7 Future work

In the thickness optimization formulation, there are several other features that should be introduced, e.g., displacement constraints on selected nodal degrees of freedom and a generalized stop criterion.

Furthermore, it is possible to introduce a parameter that defines the range of the element thickness for each individual finite element and, thereby, also an individual updating of the parameters for each finite element. For instance, by adding a memory to the optimization process, the range parameter can be reduced for an element thickness if that thickness starts to oscillate. Also the problems used in this paper were mainly tension/compression problems, although with a nonlinear material model. However, the thickness approach may also be useful in a situation where, e.g., buckling is at hand. Therefore, investigation of how to utilize this method in a buckling problem should be investigated.

The influence of the volume constraint on the optimal topology in a dynamic loadcase needs to be further analyzed.

Acknowledgement This project has been funded by VINNOVA/ Crashworthiness. We appreciate the assistance of Ms. Celine Petit in the initial phase of this investigation.

References

Bendsøe MP, Sigmund O (2003) Topology optimization: theory, methods and applications. Springer, Berlin Heidelberg New York

Borrvall T (2001) Topology optimization of elastic continua using restriction. Arch Comput Methods Eng 8:351–385

Borrvall T, Petersson J (2001) Topology optimization using regularized intermediate density control. Comput Methods Appl Mech Eng 190:4911–4928

Bruns TE, Sigmund O (2004) Toward the topology design of mechanisms that exhibit snap-through behavior. Comput Methods Appl Mech Eng 193:3973–4000

Buhl TE, Pedersen CBW, Sigmund O (2000) Stiffness design of geometrically nonlinear structures using topology optimization. Struct Multidiscipl Optim 19:93–104

Ebisugi T, Fujita H, Watanabe G (1998) Study of optimal structure design method for dynamic nonlinear problem. JSAE Rev 19:251–255

Eschenauer HA, Olhoff N (2001) Topology optimization of continuum structures: a review. Appl Mech Rev 54:331–390

Fredricson H, Johansen T, Klarbring A, Petersson J (2003) Topology optimization of frame structures with flexible joints. Struct Multidiscipl Optim 25:199–214
Hallquist JO (1998) LS-DYNA theoretical manual. Livermore Software Technology Company, Livermore
Hallquist JO (2002) LS-PRE/POST v1.0. Livermore Software Technology Company, Livermore
Hallquist JO (2004) LS-DYNA user's manual v. 970. Livermore Software Technology Company, Livermore
Pedersen CBW (2003) Topology optimization design of crushed 2Dframes for desired energy absorption. Struct Multidiscipl Optim 25:368–382
Pedersen CBW (2004) Crashworthiness design of transient frame structures using topology optimization. Comput Methods Appl Mech Eng 193:653–678
Petersson J (1998) Some convergence results in perimeter-controlled topology optimization. Comput Methods Appl Mech Eng 171: 123–140
Rainsberger R (2001) TrueGrid Manual v2.1.0 XYZ. Scientific Applications, Livermore
Sigmund O (2001a) Design of multiphysics actuators using topology optimization—Part I: one-material structures. Comput Methods Appl Mech Eng 190:6577–6604
Sigmund O (2001b) Design of multiphysics actuators using topology optimization—Part II: two-material structures. Comput Methods Appl Mech Eng 190:6605–6627
Sigmund O, Petersson J (1998) Numerical instabilities in topology optimization: a survey on procedures dealing with checkerboards, mesh dependencies and local minima. Struct Optim 16:68–75
Soto CA (2001a) Optimal structural topology design for energy absorption: a heuristic approach. In: Proceedings of the 2001 ASME design engineering technical conference, Pittsburgh, PA, USA, DETC01/DAC-21126, 9–12 September 2001
Soto CA (2001b) Structural topology optimization: from minimizing compliance to maximizing energy absorption. Int J Veh Des 25:142–163
Soto CA (2002) Applications of structural topology optimization in the automotive industry: past, present and future. In: Fifth world congress on computational mechanics, Vienna, Austria, 7–12 July 2002
Torstenfeldt B (1998) Trinitas developer studio. http://www.solid.ikp.liu.se/~botor/TRINITAS_Learning_Studio.html