

Topology Design using LS-TaSC™ Version 2 and LS-DYNA®

Willem Roux

Livermore Software Technology Corporation, Livermore, CA, USA

Abstract

This paper gives an overview of LS-TaSC version 2, a topology optimization tool using LS-DYNA for the analysis of nonlinear structural behavior. The focus is on its capabilities, current development directions, and integration into an industrial design environment. Examples of using the new developments such as global constraints, geometric definitions such as symmetry and casting directions, and shells are given.

1 Overview

The goal of topology optimization is to find the shape of a structure with the maximum utility of the material. For dynamic problems like crashworthiness simulations, this is achieved by designing for a uniform internal energy density in the structure while keeping the mass constrained.

The overall LS-TaSC [1] process consists of (i) the design problem definition, (ii) performing the design optimization iteratively using LS-DYNA [2], and (iii) post-processing the results.

The topology design problem is defined by (i) the allowable geometric domain, (ii) how the part will be used, and (iii) properties of the part such as manufacturing constraints. Additionally, you have to specify methodology requirements such as termination criteria and management of the LS-DYNA® evaluations. The information is grouped using the following three headings:

- *Cases* These store the load case data such as, the LS-DYNA® input deck and executable to use. The *Cases* data therefore contain the information on how to simulate the use of part.
- *Part* The rest of the problem definition data such as the part ID, geometric definitions such as being an extrusion, and the desired mass is given here.
- *Constraints* These constraints on the global behavior of the structure, such as the stiffness and the compliance.
- *Completion* These are methodology data such as the convergence criterions.

The initial parts specify the design domain – the optimum parts computed will be inside the boundaries delimited by the initial parts. The parts must be modeled using solid or shell elements.

The part may contain holes; a structured mesh is accordingly not required and there is no node or element numbering convention as in other approaches. Geometry constraints such as being an extrusion or a casting direction may be specified.

The use of the part is described by LS-DYNA input deck. The design process aims for a uniform internal energy density in the structure as computed by LS-DYNA using this input deck.

The final shape of the part is described by the subset of the initial elements used. The use of an element is controlled by changing the amount of material in the element. This is achieved by assigning a design variable to the density of each element. The material is parameterized using a so-called *density approach*. In this approach, a design variable is directly linked to the individual material element such that each cell has its own material model. The design variable x , also known as relative density, varies from 0 to 1 where 0 indicates void and 1 represents the full material. The material properties corresponding to the values of design variables are obtained using an appropriate interpolation model as described in the manual [1].

2 Methodology

The typical goal of topology optimization is to obtain maximum utility of the material. Obtaining uniform internal energy density in the structure is used as the objective for optimization. This follows the formulation proposed by Patel [4], with the resulting implementation being similar to the fully-stressed design and uniform strain energy density approaches (Haftka and Gurdal [5], Patnaik and Hopkins [6]).

The optimization problem is formulated as,

$$\begin{aligned} \min_x \quad & \sum_{i=1}^N \sum_{j=1}^L (w_j U_j(x_i) - U_j^*), \\ \text{subject to:} \quad & \sum_{i=1}^N \rho(x_i) V_i \leq M^* \\ & C_j^l \leq C_j \leq C_j^u, \quad j = 1, 2, \dots, J \\ & x_{\min} \leq x_i \leq 1.0. \end{aligned} \quad (7)$$

where U represents the internal energy density of the i^{th} element, V_i is the volume of i^{th} element, U^* represents internal energy density set point, and C_j is the j^{th} constraint. There are L load cases with a total of J constraints. The superscripts ' l ' and ' u ' represent lower and upper bounds on the constraints, respectively.

The change in the design variable of i^{th} variable (Δx_i) is computed as,

$$\Delta x_i^t = K(U_i^t - U^*)/U^*.$$

where K is a scaling factor and U^* denotes the internal energy density set point. The design variable is updated as,

$$x_i^{t+1} = x_i^t + \Delta x_i^t.$$

3 Control Parameters

The algorithm, and therefore the final design, can be controlled using some parameters:

- **Mass Fraction** The optimal design is created by deleting a fraction of the original part. The *Mass Fraction* parameter controls the amount of mass preserved. The effect of different values of this parameter is shown in Figure 1 using the default value of the other parameters.
- **Convergence Tolerance Parameter** The analysis is terminated when the change in the *Mass Redistribution* (view this history in the viewer panel) for the iteration is less than the *Convergence Tolerance*. The *Mass Redistribution* can be interpreted as the portion of material moved in an iteration; if the *Mass Redistribution* is 1.0 (not actually possible), then all possible mass have been redistributed.
- **Proximity Tolerance Parameter** This parameter describes a distance controlling the neighborhood size of the elements. The design variable at an element is updated using the result at the element averaged together with that of its neighbors. Smaller values of this parameter yield finer-grained structures as show in Figure 2.

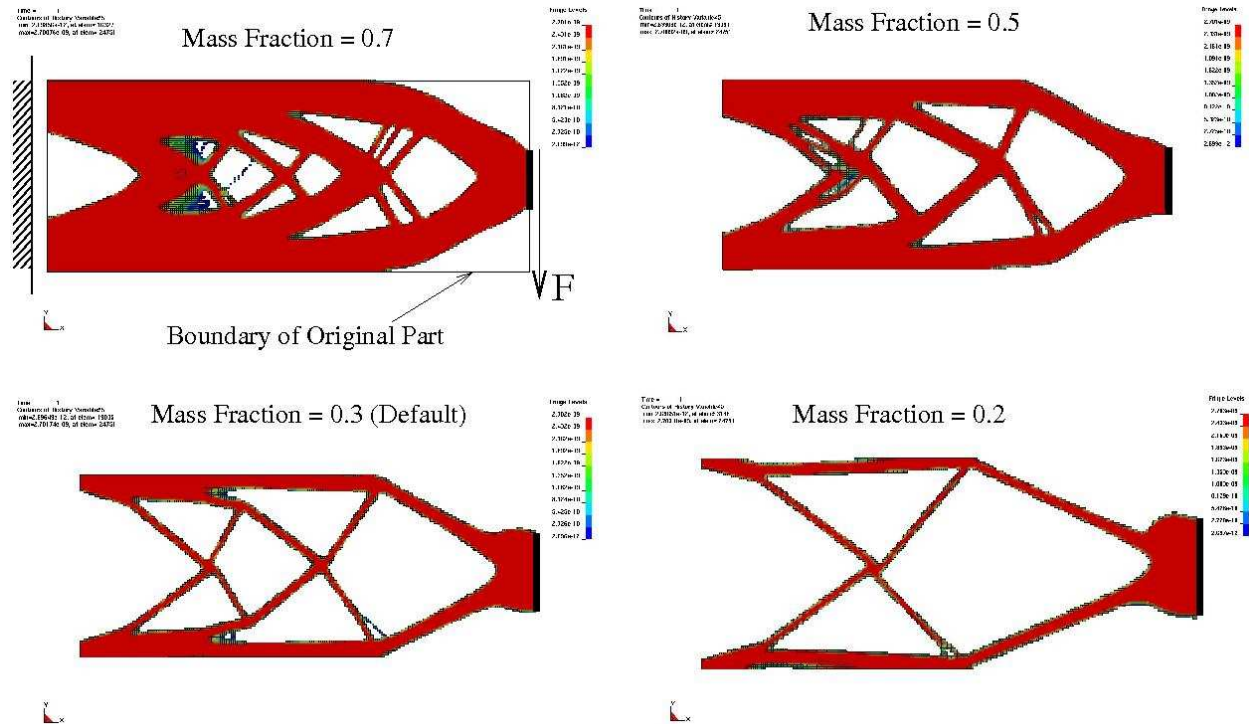


Figure 1 : The effect of the mass fraction bound. Different designs created using different mass fractions are shown. The mass fraction is the fraction of original mass that must be kept. So for a mass fraction of 0.3, the software will remove 70% of the original structure.

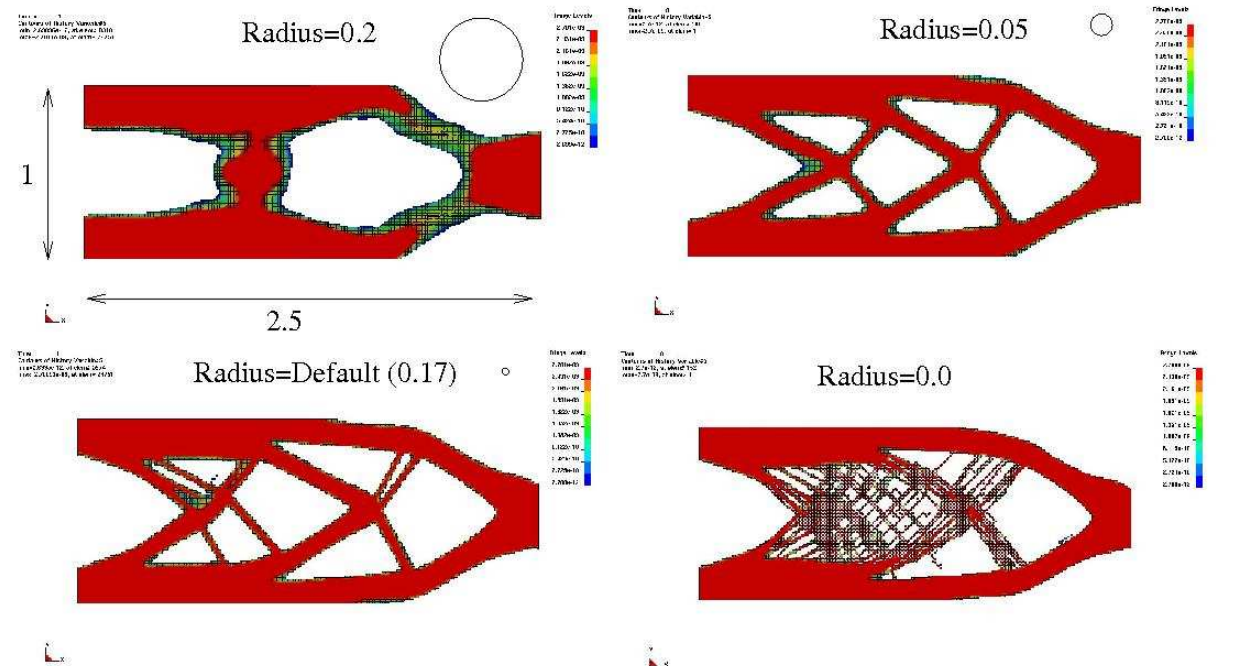


Figure 2 Effect of the proximity tolerance parameter. The mesh is 100 x 250. The small circles show the neighborhood size. A parameter value of zero results in a checkerboard pattern.

4 New capabilities in version 2

The following new capabilities were added for version 2.

- **Global constraints** The global constraints are used to constraint responses caused by the whole part, such as a displacement or reaction force. This is actually a method of computing the mass fraction, because the mass of the structure is increased or decreased until the constraints are satisfied.
- **Shells** The thickness of shells can be redesigned to have a uniform distribution of the internal energy density. Unlike for the solids, a 0/1 behavior is not enforced, and the goal is therefore not to create a new geometry, but to find a thickness distribution.
- **Geometric constraints and multiple parts** These are constraints on the final geometry allowing practical parts, which can be manufactured, to be designed. The GUI interface showing the definition of multiple parts and geometric constraints is shown in Figure 3. The different types of geometry constraints are:
 - extrusions
 - casting, one sided
 - casting, two sided and
 - symmetry.

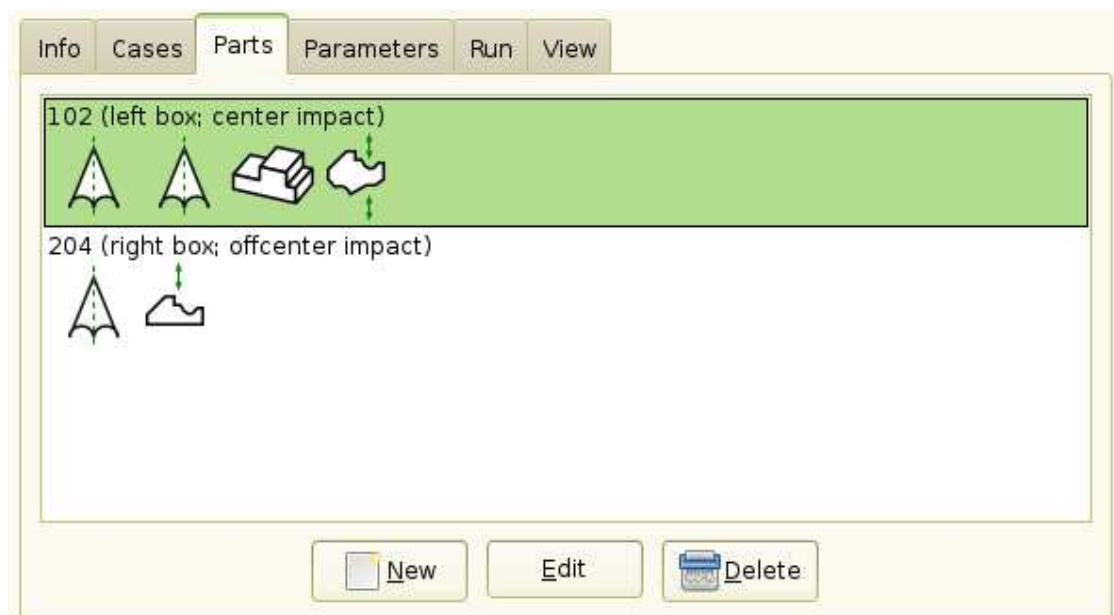


Figure 3 Different parts, each with multiple geometry constraints, can be designed simultaneously. This allows parts with complex geometries to be designed.

5 Collaboration with other CAE Companies

Beta CAE Systems S.A., the author of the ANSA preprocessor, is looking into CAE work flow procedures incorporating LS-TaSC. In addition, Detroit Engineered Products, author of MeshWorks Morpher, is investigating methods of using their mesh refinement technologies. Contact these parties for more information.

6 Examples

6.1 Example with global constraints together with multiple geometry definitions

The problem as shown Figure 4 consists of a beam being subjected to two impact load cases as shown.

The problem has two symmetry constraints and an extrusion constraint. It is required to be an extrusion in the z-direction and to be symmetric about both the YZ and ZX plane as shown in the figure.

The problem also has two global constraints: for each load case it is required that the displacement is less than 100 units.

The resulting optimal design is shown in Figure 5 with the histories of the constraint values for the iterations as shown in Figure 6.

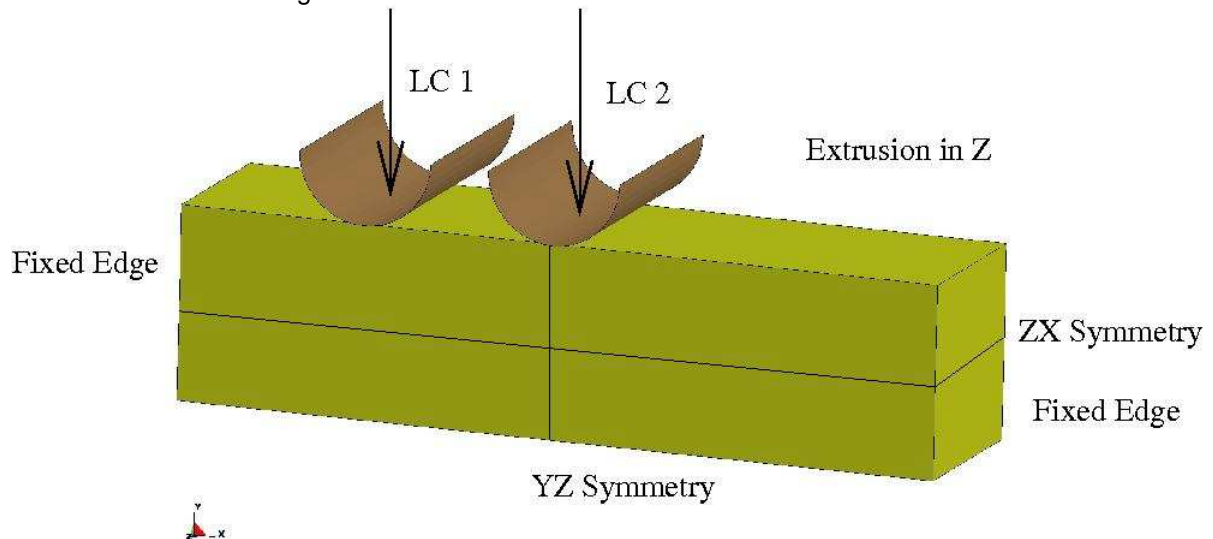


Figure 4 Design problem with two load cases and three geometric constraints

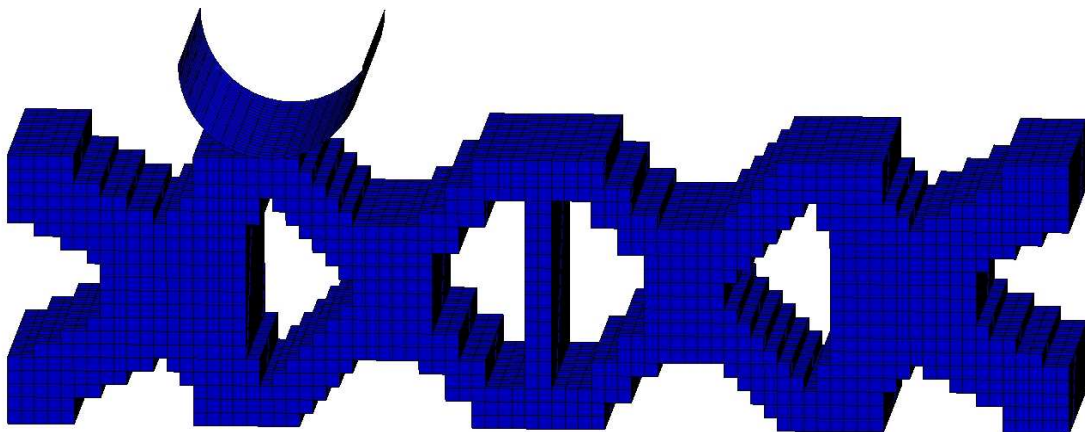


Figure 5 Optimal design. The part is an extrusion symmetric about two planes.

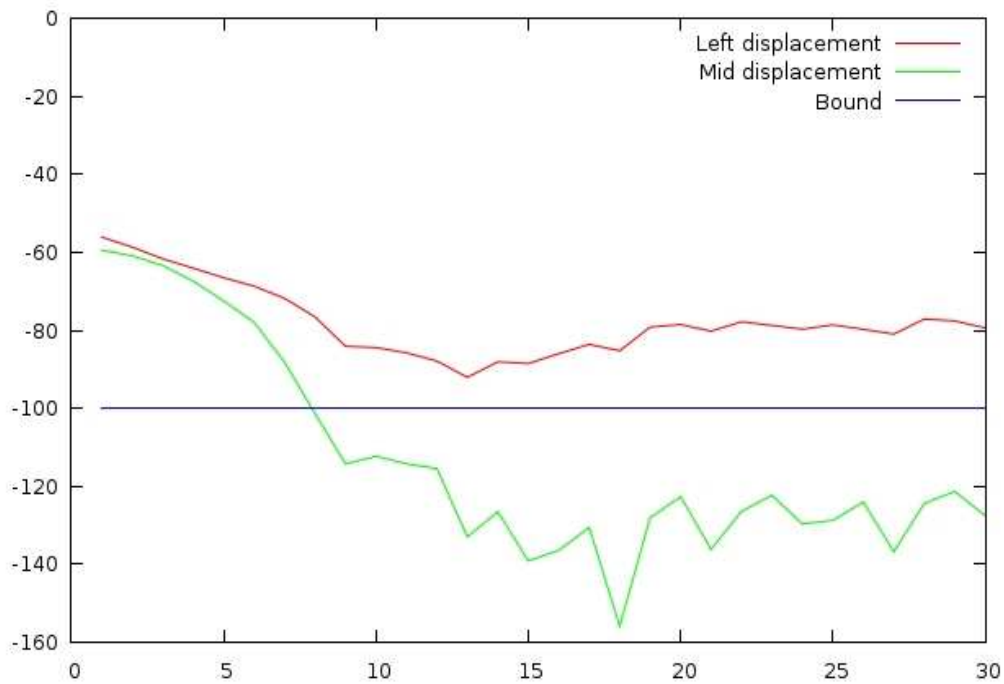


Figure 6 Constraint histories. The final design required a trade-off between the two constraints. This is because the goal of topology is to get a good concept design, not to satisfy the constraints exactly.

6.2 Casting example

This example is a solid part to be manufactured as a casting, which was accordingly imposed as a casting geometry definition. Weight (material) was to be removed from the structure to obtain the best use of the material. The procedure accordingly computed an optimal design by strengthening stressed regions of the structure and removing redundant material. Alternative, this design procedure can also be viewed as computing a structure with the best load path for the given structural use.

The geometry and loading conditions for this component are shown in Figure 7. The FE model has about 60 000 elements and a single linear implicit load case as shown was considered.

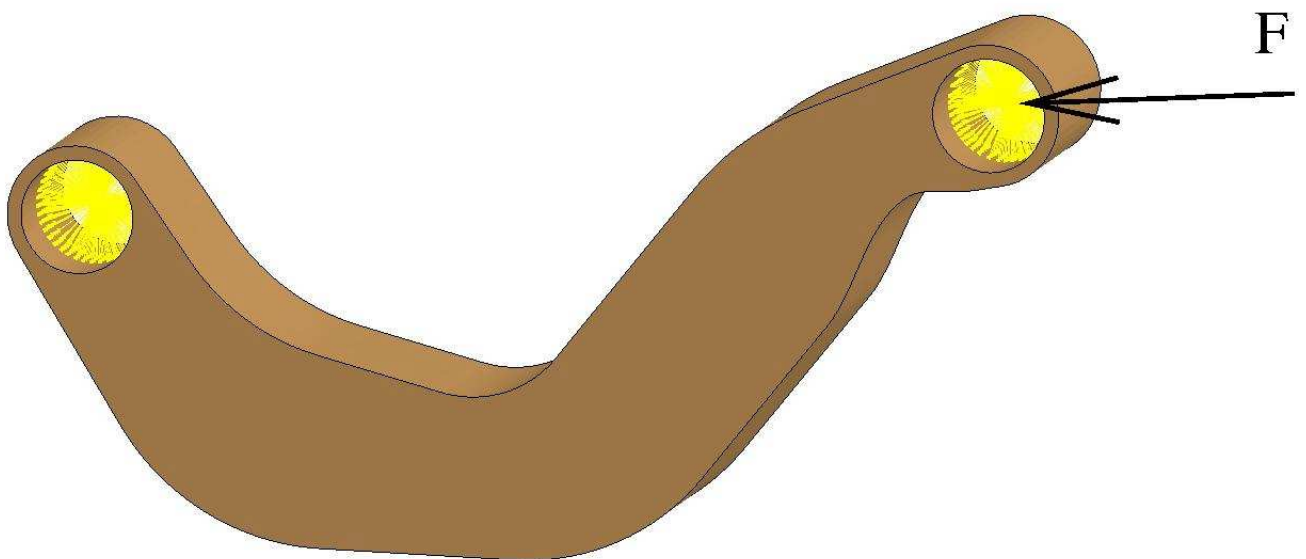


Figure 7: The initial geometry and loading conditions.

The convergence history for the multiple-load example is shown in Figure 8. The initial and final structures are shown in Figure 9, while final design is shown in Figure 9. The mass fraction specified in the history is the amount of material that the user specified should be kept – in this case half (a mass fraction of 0.5) of the structure was scheduled to be retained. The element fraction used is the fraction of the original number of elements in the part used at any point in the design cycle – this value will therefore converge to close to the requested mass fraction. The mass redistribution is the fraction of the overall mass moved around in the design cycle – a small number indicates convergence of the procedure.

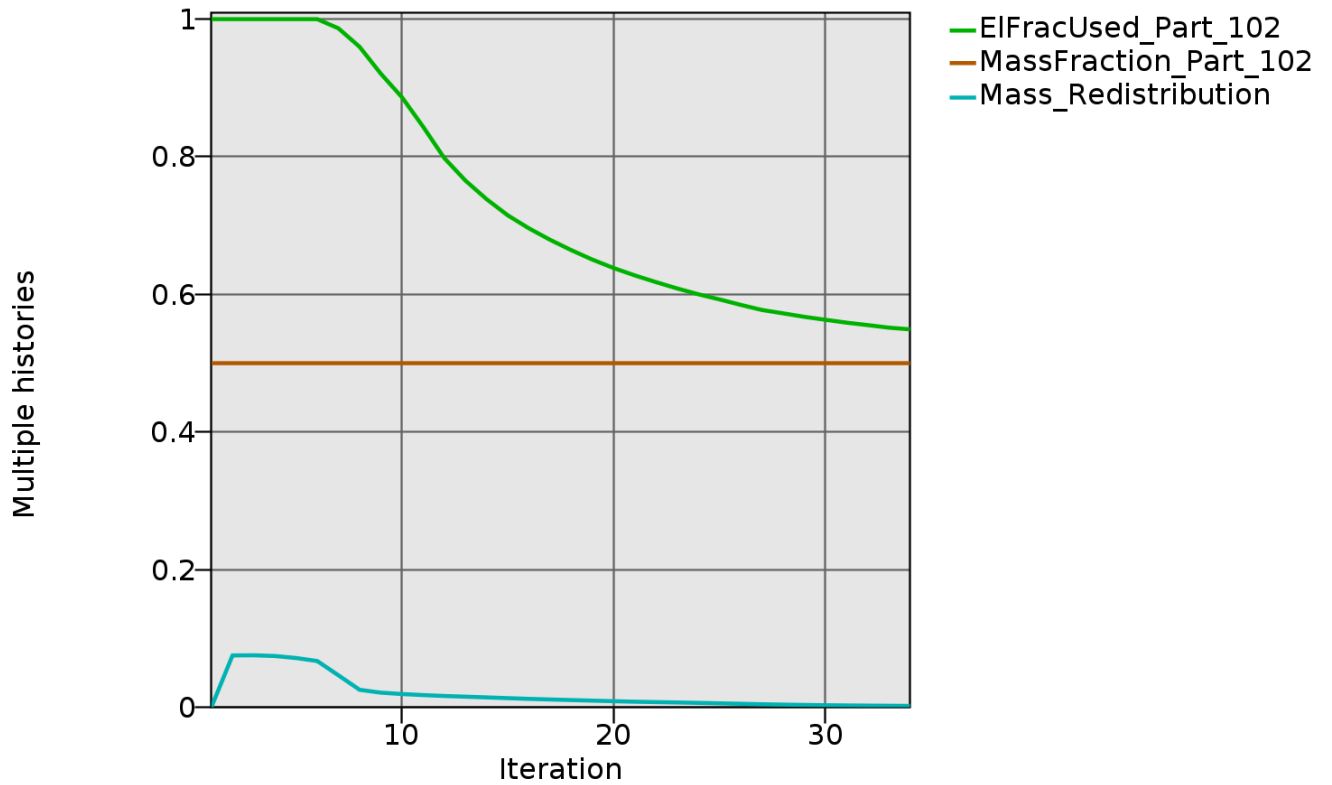


Figure 8: Convergence history. The problem is converged when the mass distribution is nearly zero. At that time the fraction of elements used is also close the requested mass fraction.



Figure 9: Final design for casting problem. The outline of the original design is also shown.

6.3 Shell example

The geometry and loading conditions for the shell example are shown in Figure 10. For a shell problem such as this, the thickness of each shell element is a design variable.

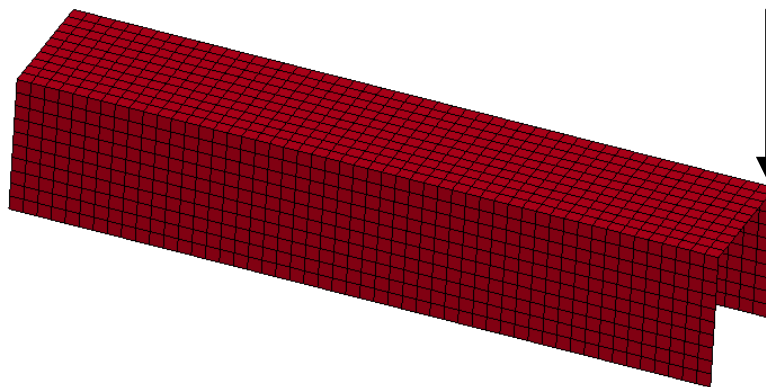


Figure 10 The geometry and loading conditions of the shell example. The left side is built-in, while a downward load is applied to the right, back corner.

The final geometry with the final thickness is shown in Figure 11. Shell elements with a thickness of less than 5% of the starting design were deleted in the final design. The final part is stronger on the side where the load is applied and at the left rear where the bending moment is the largest.

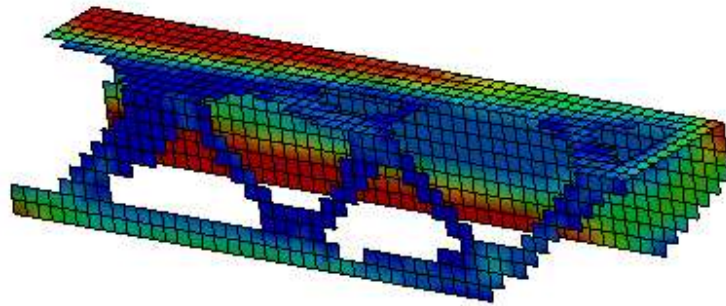


Figure 11: The final geometry and thicknesses for the shell problem

7 Current Development

Current improvements and extensions are:

- Forging geometry definition. This is similar to a casting definition, except holes are not allowed to extend through the structure; instead a web of a certain thickness must remain. See Figure 12 for an example.
- Frequency constraint. The structure will be forced to meeting basic frequency constraints.
- Support of more LS-DYNA material models. The program used to only consider only *MAT_PIECEWISE_LINEAR_PLASTICITY. The user will be able to use *MAT_ELASTIC as well as others material models as requested by industry.
- Different fields for the objective. Currently the design is done to obtain a uniform internal energy density. This extension allows plastic strains and Von Mises stresses also to be considered.

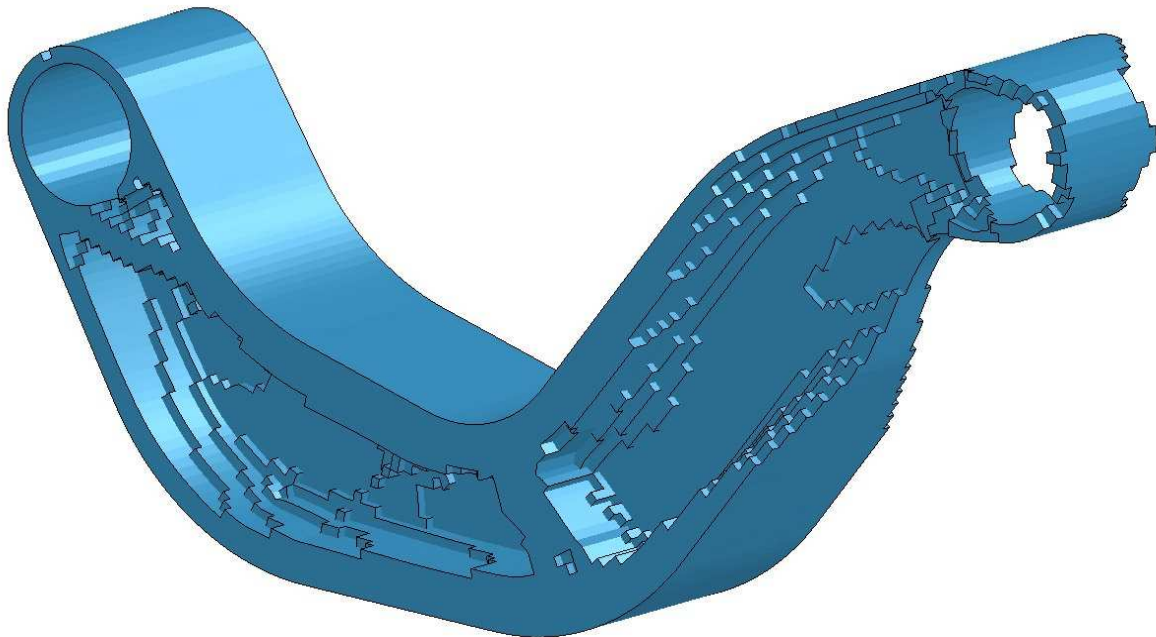


Figure 12 Forging definition design. Notice the web that is required to use a forging manufacturing process.

8 Summary

LS-TaSC computes the shape of a structure with the maximum utility of the material. It has been developed for non-linear structures analyzed in an industrial environment. The tool is also suitable for

large linear problems. This tool has been extended to shells, global constraints, multiple parts, symmetry, and extrusion constraints.

References

- [1] Livermore Software Technology Corporation, LS-TaSC™: A Topology and Shape Computations for LS-DYNA, User's Manual, Version 1.0, Livermore Software Technology Corporation, Livermore, CA, 2009.
- [2] Hallquist JO. LS-DYNA theoretical manual, Livermore Software Technology Corporation, Livermore, CA, 1998.
- [3] Goel T, Roux WJ, and Stander N. A Topology Optimization Tool for LS-DYNA users: LS-OPT/Topology. 7th European LS-DYNA Users Conference, Salzburg Austria, 14-15 May 2009.
- [4] NM Patel, *Crashworthiness Design Using Topology Optimization*, PhD thesis, University of Notre Dame, 2004.
- [5] RT Haftka, Z Gurdal, MP Kamat, *Elements of Structural Optimization*, Kluwer Academic Publishers, Dordrecht, The Netherlands, 2nd ed., 1990.
- [6] SN Patnaik, DA Hopkins, Optimality of Fully-Stressed Design, *Computer Methods in Applied Mechanics and Engineering*, 165, 215-221, 1998.