

LS-OPT®/Topology Version 1

Willem Roux†, Tushar Goel†, David Björkevik‡

†Livermore Software Technology Corporation, ‡Engineering Research AB

Abstract

This paper presents LS-OPT/Topology, a new topology optimization tool. Topics such as its capabilities, current development directions, and integration into an industrial design environment are discussed.

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-OPT/Topology [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.

Design Problem Definition

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.
- *Problem* The rest of the problem definition data such as the part ID and extrusion definition is given here.
- *Method* These are methodology data such as the convergence criterions.

The Design Part

The initial part specifies the design domain – the optimum part computed will be inside the boundaries delimited by the initial part. The part must be modeled using *MAT_PIECEWISE_LINEAR_PLASTICITY using only eight-noded solid 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. Design constraints such as being an extrusion 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 theoretical manual.

Control 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.

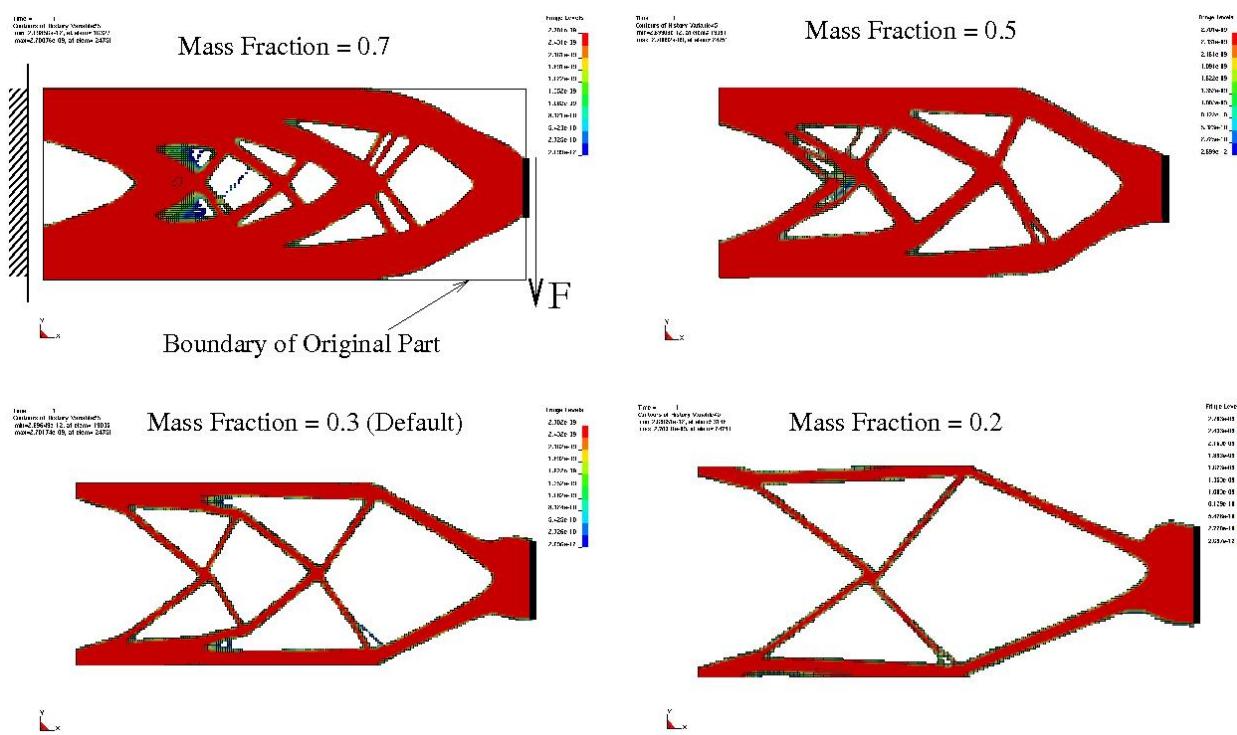


Figure 1 : The effect of the mass fraction bound. Different designs created using different mass fractions are shown. The mass fraction is the fraction of original mass that must be kept. So for a mass fraction of 0.3, the software will remove 70% of the original structure.

Convergence Tolerance Parameter

The analysis is terminated when the change in the *Density Redistribution* (view this history in the viewer panel) for the iteration is less than the *Convergence Tolerance*. The *Density*

Redistribution can be interpreted as the portion of material move in an iteration; if the *Density Redistribution* is 1.0 (not actually possible), then all possible mass have been redistributed.

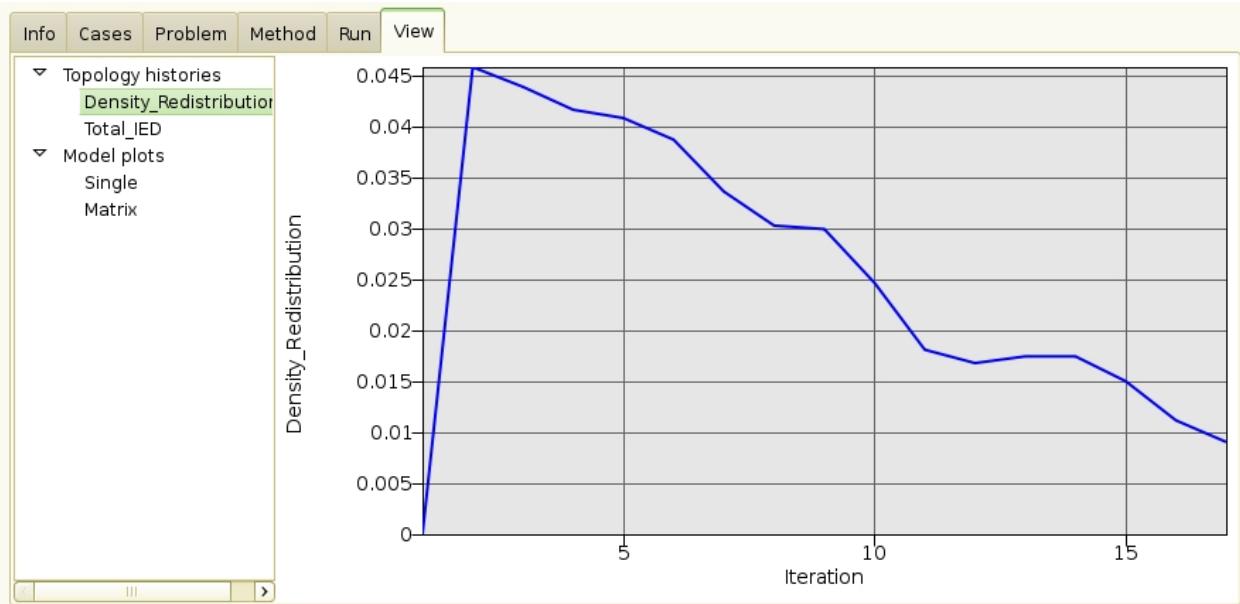


Figure 2 The Density Redistribution at every iteration. The optimization is terminated when this value is less than the convergence tolerance. The parameter indicates which percentage of the geometry have altered during a design iteration.

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 3.

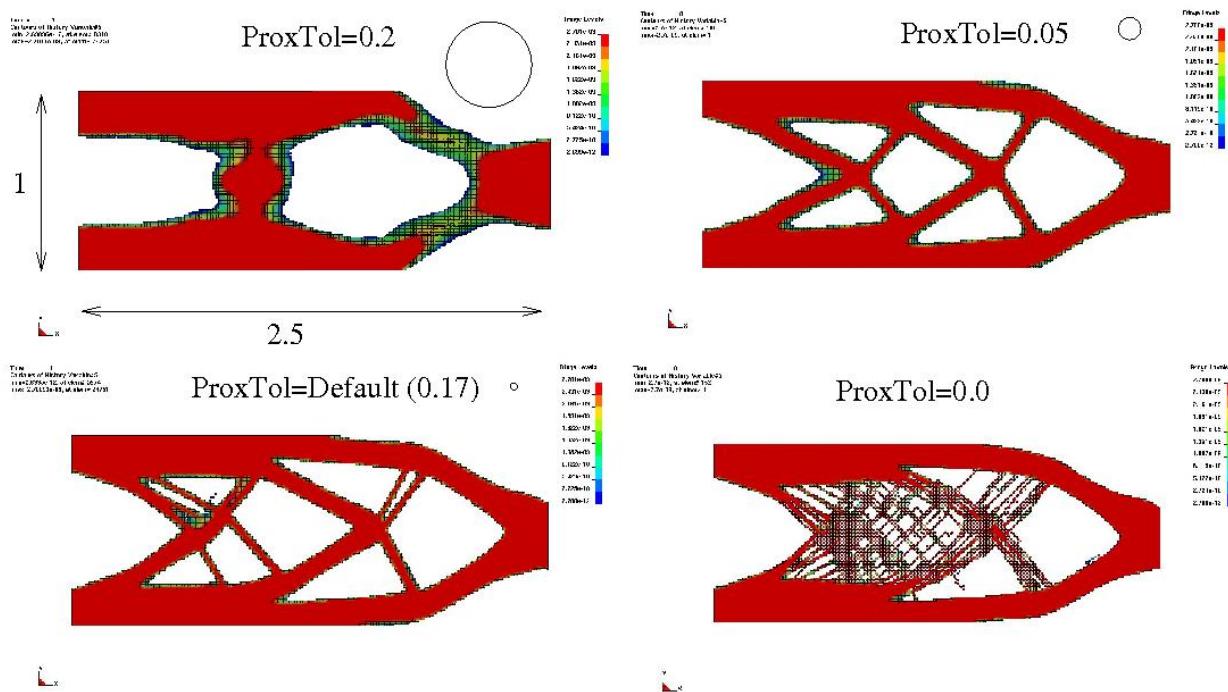


Figure 3 Effect of the proximity tolerance parameter. The mesh is 100 x 250. The small circles show the neighborhood size. A parameter value of zero results in a checkerboard pattern.

FE Analysis and Distributed computing

The designs are analyzed using LS-DYNA®. More than 10^6 elements may be used.

One can take advantage of multiple processors using the MPP version of LS-DYNA®, and in addition the computations can be distributed using queuing systems.

Post-processing

Two types of post-processing are possible:

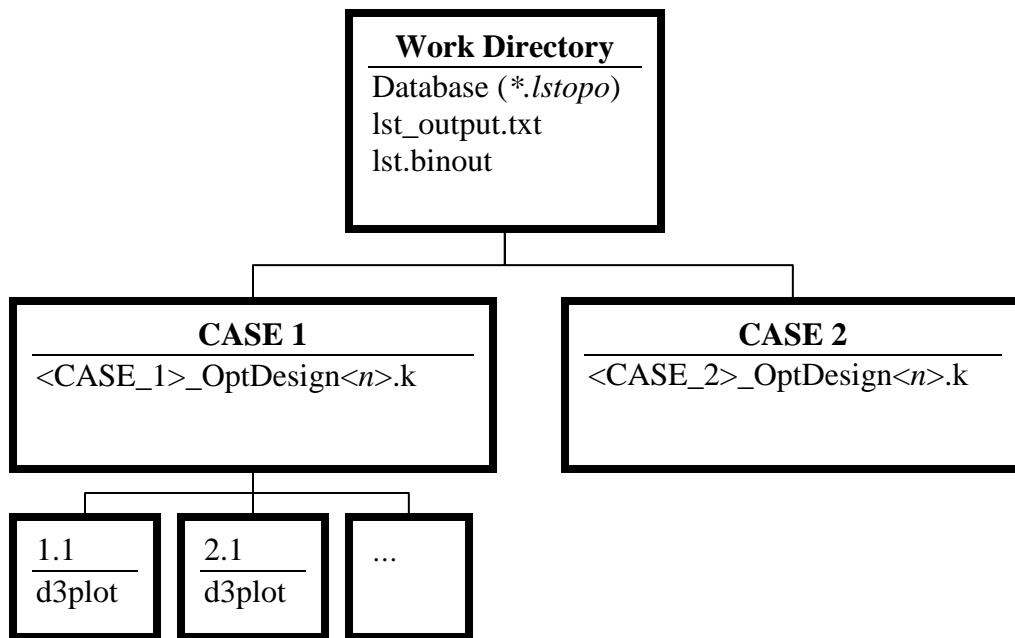
- History plots of scalar quantities, such as the Energy Redistribution as shown in Figure 1 and Figure 5.
- FE plots in LS-PREPOST®. This FE plots show the design evolution as illustrated in Figure 6 and Figure 7.

Databases and Files

The important files and directories are shown in the figure below. Four files are important to know about:

- The project database
- The project results in the *lst.binout* binary file

- The optimal design in the case directory
- The d3plot files in the run directory inside the case directory



Scripting

Scripting was added for advanced users and support. Normal use is through the user interface, but advanced users can write their own programs, while support can use this to debug problems. The scripting can be used for task such as:

- Convert the data in the database into any other format.
- Write programs calling the topology in an iterative fashion or user design procedures.
- Run the topology without the GUI, even from within another program.

The scripting language is based on the C computer language syntax. It is already used in LS-OPT and LS-DYNA.

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-OPT/Topology. In addition, Detroit Engineered Products, author of MeshWorks Morpher, is investigating methods of using their mesh refinement technologies. Contact these parties for more information.

Current Development

The next version will address the following two topics:

- Constraints such as maximum displacements.
- Design of shells structures.

Various smaller items from customer feedback will also be included.

Nonlinear Example

The example demonstrates the simulation with multiple load cases.

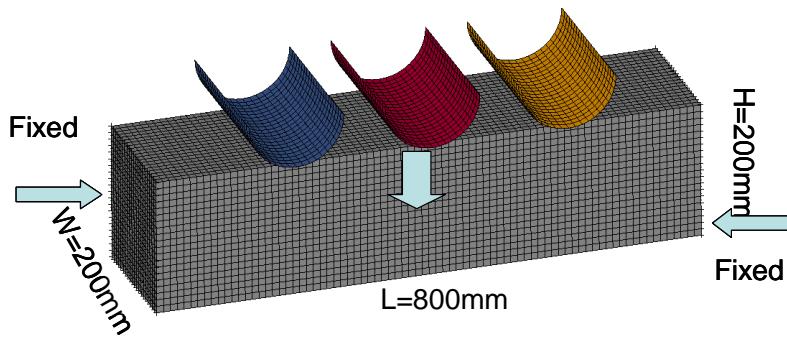


Figure 4 The geometry and loading conditions of the multiple load case example.

The geometry and loading conditions for the example are shown in Figure 4. This is fixed-fixed beam with three loads. The design part was meshed with 10mm^3 elements.

The three load cases were identified according to the location of the pole hitting the beam. Side load cases were assigned a unit weight and the center load was assigned a weight of three units. All cases were simulated using a cluster with a queuing system and eight processors per load were allocated. All simulations were run simultaneously. The desired mass fraction for this example was 0.3 and a maximum of 100 iterations were allowed.

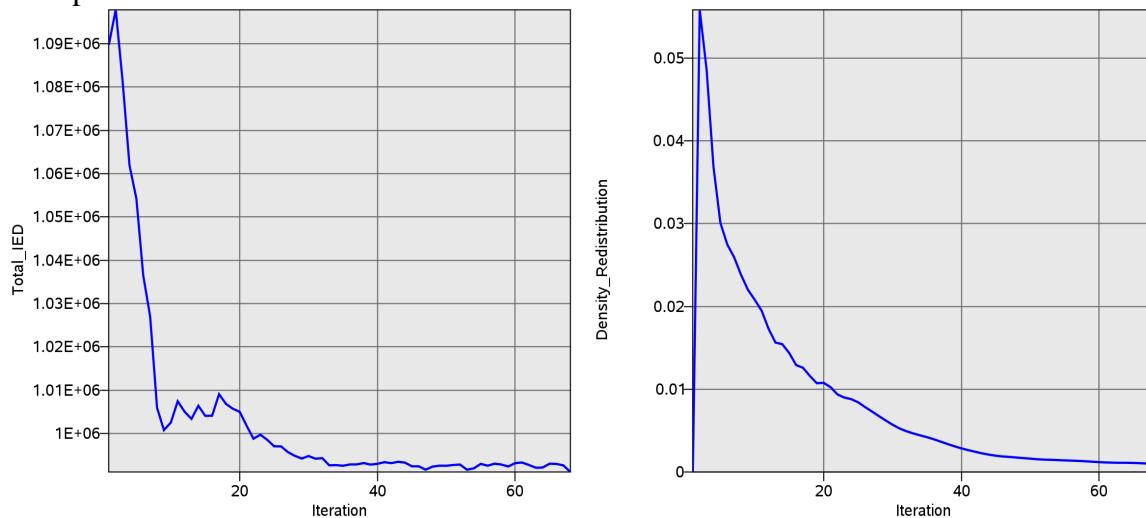


Figure 5: Convergence history for multiple-load case example.

The convergence history for the multiple-load example is shown in Figure 5. The simulation converged after 67 iterations, though minuscule changes were noted after 40 iterations. A monotonic reduction in the change in topology, characterized by the normalized *Density_Redistribution* graph, was observed. The final structure absorbed approximately 8% less total internal energy.

The initial and final structures are shown in Figure 6. The final structure evolved in a tabular structure with the two cross-members as legs. The structure had more material in the center section due to very high importance assigned to the center weight. There were many cavities in the structure such that the final structure could be considered equivalent to a truss-like structure as one would expect.

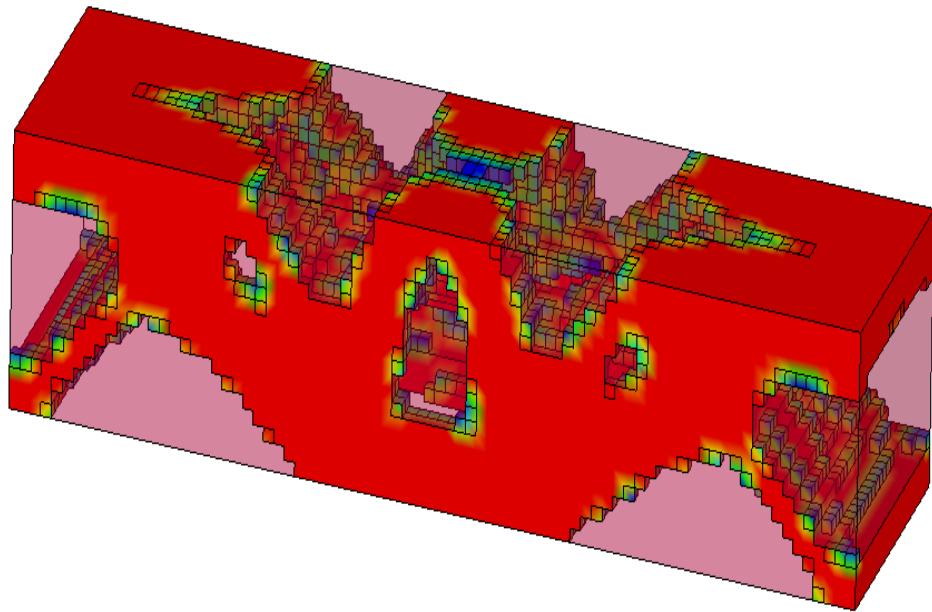


Figure 6: Final design with density contours superimposed on the initial design.

The evolution of the topology under multiple loading conditions is shown in Figure 7. While the final form of the structure was largely evolved by the 28th iteration (row 2, column 1), the material was re-distributed to remove the low density material and evolve a largely 0/1 (no material or full density material) structure.

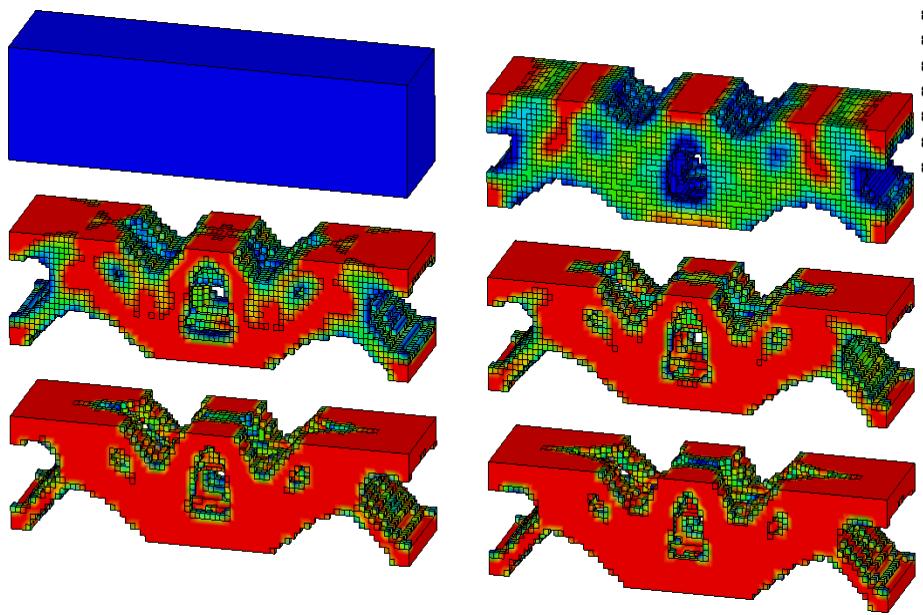


Figure 7: Evolution of the geometry for multiple-load case structure.

Summary

LS-OPT/Topology 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. LSTC will continue to extend this tool.

References

- [1] Livermore Software Technology Corporation, *LS-OPT/Topology: A Topology Optimization Tool 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.