Getting Started with LS-TaSC v3.0

What is Topology Optimization?

Topology optimization is a structural design optimization technique for distributing material efficiently across the design domain by changing the topology of the design for a given set of loading and boundary conditions such that, the resulting design has better performance targets in terms of improved stiffness, and reduced mass.

Even though it is considered to be more complex compared to shape and sizing optimization techniques, topology optimization has become an integral part of product design and development process. A topologically optimized structure is intended to demonstrate better or at least similar structural performance relative to its baseline design at reduced mass. Therefore, it has major applications in aircraft, automotive and other structural design industries were mass of the structure is critical in its design.

In topology optimization, the overall topology of the structure is varied by removing unwanted material from the structure such that the final design is lighter without compromising the performance characteristics.

How is it different from sizing and shape optimization?

In a sizing optimization problem, the design variables are usually geometrical parameters such as length, width or thickness of the part being optimized. The optimization process involves search of optimum values for these parameters such that design objectives and constraints are met. LS-OPT design optimization tool can be used for sizing or parameter optimization problems.

Similarly, shape optimization deals with optimizing the overall shape of the structure such that the optimum design results in a structure with uniform stress distribution eliminating the stress concentration. This feature is available in LS-TaSC starting from version 3.0.

Topology Optimization in LS-TaSC 3.0

LS-TaSC is a topology and shape computation tool of LS-DYNA suite developed by Livermore Software Technology Corporation. The initial implementation of optimization process involved a heuristic optimization method developed by University of Notre Dame but was later modified and developed using more general approaches.

Key Concepts

Design Objective:

In LS-TaSC, the goal is to obtain a structure with uniform energy density distribution to suit crashworthiness applications where internal energy absorption of the design parts is important.

Design Variables:

In LS-TaSC, the design variables depend on the type of elements being used for the structure. For example, element thickness is considered as design variables for shell elements whereas, for solid elements, relative density of the elements is treated as design variables. A user-defined lower bound allows deletion of the elements violating this bound.

Design Constraints:

A user defined mass fraction is provided as input for the part being optimized and it is treated as a design constraint such that unwanted material is removed during the optimization process. The user-defined mass fraction corresponds to the amount of mass retained after the optimization process.

Along with mass fraction, stiffness and compliance of the part being optimized are considered as global design constraints. The global responses such as nodal displacements and reaction forces can be defined as stiffness and compliance based design constraints. Other LS-DYNA *binout* responses can also be defined as constraints using the *User-defined* option. It is important to note that the global constraint violation obtained in an iteration is handled by adjusting the target mass fraction of the next iteration.

Apart from global structural response constraints, geometric and manufacturing definitions such as symmetry, casting, forging and extrusion can also be initialized for a structure. The final optimum design will be in accordance with the geometric and manufacturing definitions.

Dynamic Load Case Weighing:

In case of problems involving multiple load cases, it is not desired to have a certain load case to dominate the overall topology design. This can be handled by using dynamic load weighing feature. A direct relationship between global constraints of multiple load cases can be defined and weights of the load cases are adjusted dynamically to satisfy this relationship.

Optimization Process:

The optimization process involves an iterative based technique for finding a topology of structure with reduced mass and uniform internal energy density also satisfying the global constraints. The structural design input such as geometry, material data, contacts etc are provided to LS-TaSC in the form of an LS-DYNA input file. Therefore, during each iteration, the input is rewritten by modifying the element data and LS-DYNA analysis tool is used as a solver to determine the responses associated with each topology. When convergence criteria are met, the optimum design is written as an LS-DYNA keyword file.

Convergence Criteria:

The optimization process stops when user defined maximum number of iterations or mass redistribution tolerance are met. The mass redistribution refers to the total change in the topology given by the change in design variables.

User Interface and Features

The graphical user interface (GUI) of older versions of LS-TaSC (v2.1 and older) has been enhanced for version 3.0 by integrating LS-PrePost. LS-TaSC utilizes LS-DYNA analysis tool as a solver to analyze numerous topologies obtained from iterative based optimization process. Since the design model input provided to LS-TaSC is in keyword input format of LS-DYNA, the integration of LS-PrePost facilitates a user to modify the design quickly according to the

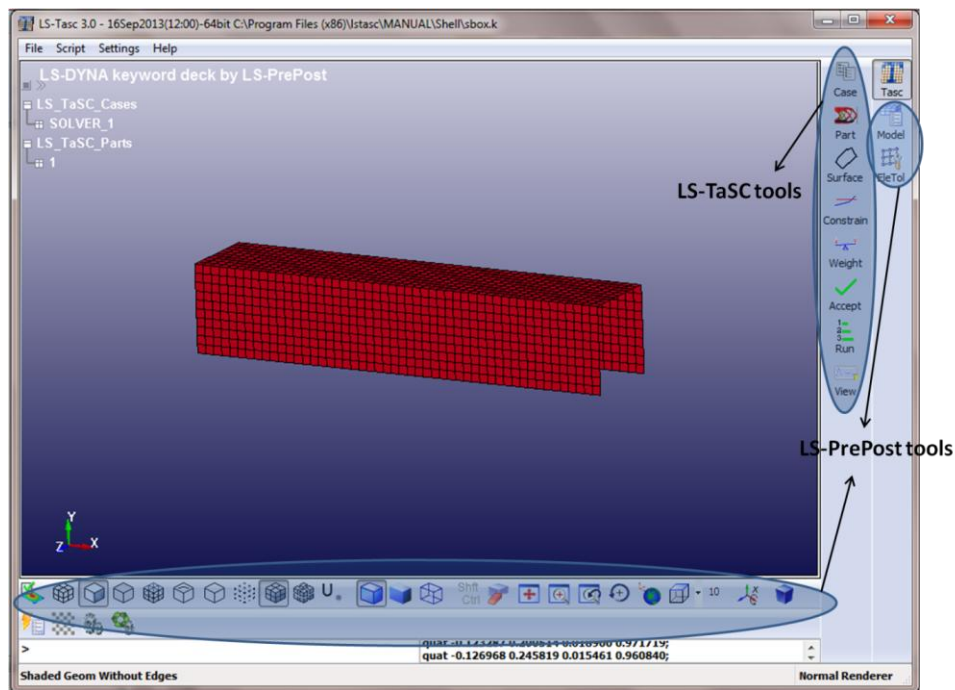optimization requirements. Figure 1 shows the user interface with both LS-TaSC and LS-PrePost tools.

Figure 1: LS-TaSC v3.0 GUI

LS-TaSC Tools

- *Case*: A *case* in LS-TaSC corresponds to the loading and boundary conditions of the structure and the resulting topology from optimization will be in accordance to this load case.

    A structure designed for a single load case can perform badly in with respect to other load cases. Therefore, multiple load case feature of LS-TaSC allows designing of structures in terms of multiple loading conditions. Any number of load cases can be added but a unique name should be provided for each case with input being the LS-DYNA keyword file. LS-DYNA executable should be selected as the solver command. Any LS-DYNA command line options such as memory/no. of CPUs can be entered directly in the command section.

    Any job scheduling options such as selection of a queuing system has to be defined in the *case* panel. *Figure 2* shows the *case* dialogue box.
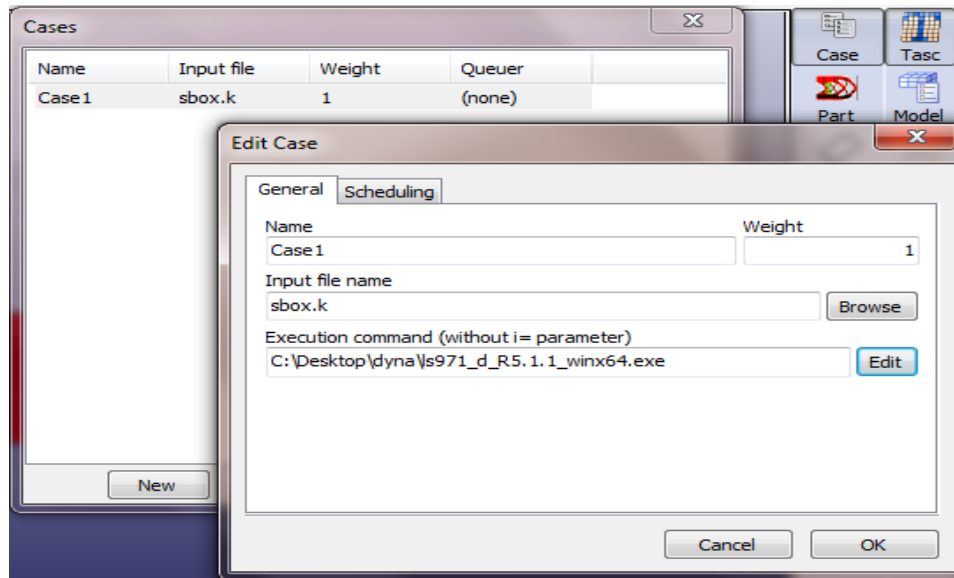
Figure 2 Case dialog box

- **Part**: The *Part* dialogue box has options to select the part to be optimized and define few optimization parameters such as mass fraction, minimum variable fraction for element deletion, element neighbor radius and geometry definitions.

    LS-TaSC does not have a limit in terms of number of parts to be optimized. Hence multiple parts can be defined and each part should be assigned optimization parameters separately. Mass fraction is the amount of mass to be retained after optimization process. A mass fraction of 0.3 indicates that LS-TaSC will try to retain 30% mass of the part. A minimum variable fraction can be specified by the user, elements with design variable values below this limit will be deleted.

    The design variables of all the elements are updated based on field variables (internal energy density) values of the neighboring elements. A virtual sphere with a radius is defined and elements within this radius are considered to be the neighboring elements (refer to LS-TaSC User's manual for more information). *Figure 3* shows the various parameters pertaining to part definition.
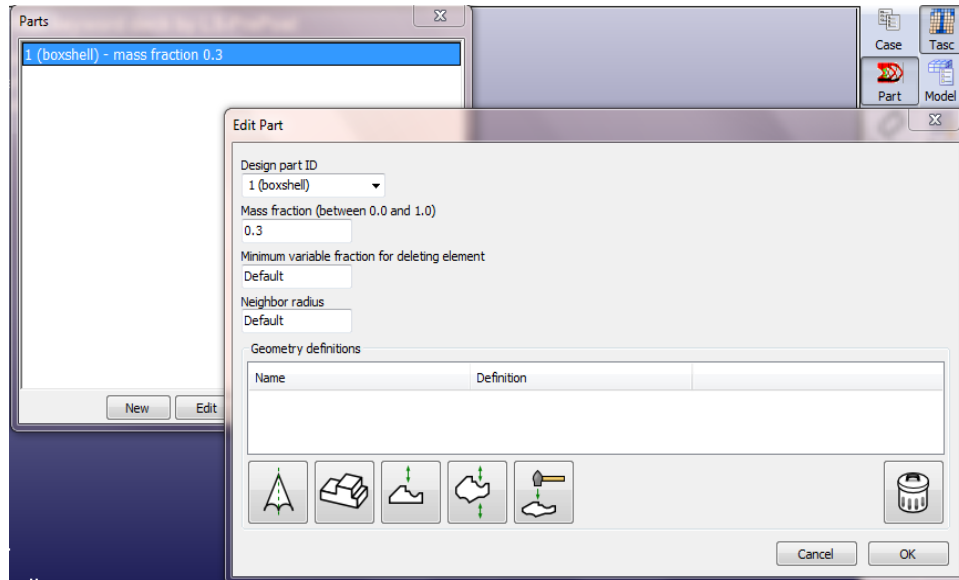
Figure 3: Part definition in LS-TaSC

Geometry and manufacturing definitions such as symmetry, casting, extrusion and forging can be defined for the part being optimized. Within casting, both one way and two way casting can be defined. There are limitations in terms of number of geometry definitions. A maximum of three geometry definitions can be assigned to each part with maximum of two casting definitions. Figure 3 shows the geometry and manufacturing definitions.
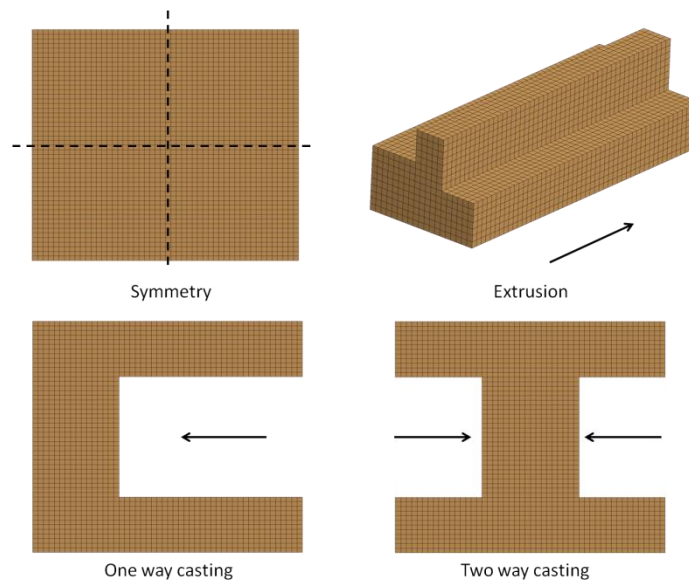

Symmetry

Extrusion

One way casting

Two way casting

Figure 4: Geometric and manufacturing definitions

- *Surface:* Surface design feature has been implemented in LS-TaSC starting from version 3.0. It optimizes the nodal locations of the selected surface to obtain a surface with uniform stress distribution. Figure shows the various parameters available for a surface design problem.
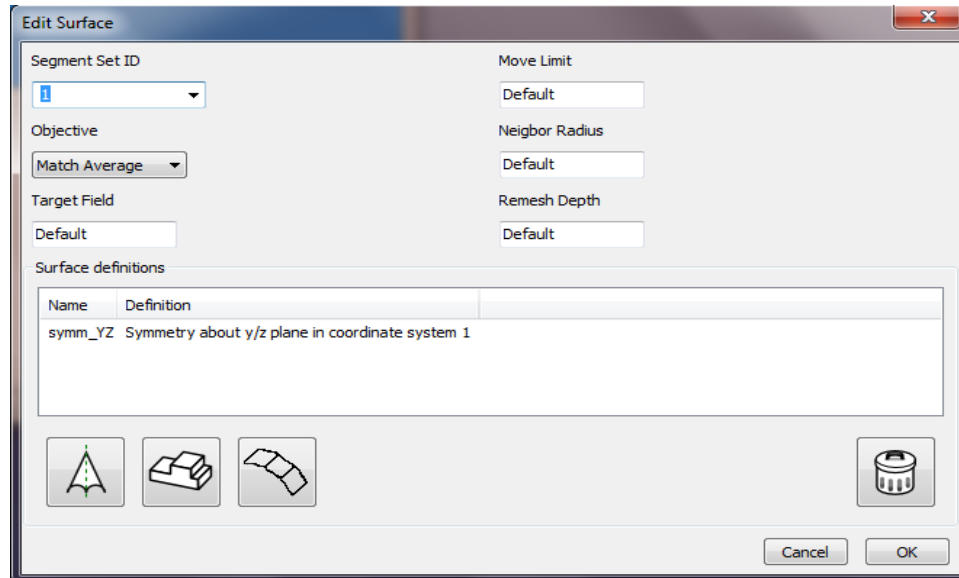
Figure 5: Surface design panel.

- *Constraints:* The constraints panel is used to define any global constraints required for the design. In the current version, displacement, reaction force and other *binout* responses of LS-DYNA analysis can be defined as constraints. Lower and upper bound fields specify the feasibility of the design. The constraints are load case specific. Therefore, appropriate load case should be selected while defining the global constraints.
- *Weights:* This panel is used to activate dynamic load weighing. A relationship between constraints of each load case can be defined in this panel. It is advised to scale the constraints by their respective upper bounds as the constraint values can differ in order of magnitude.
- *Accept:* The termination criteria for the optimization process in terms of maximum number of iterations and optimization convergence tolerance are defined in this panel.

- **Run:** This panel is used to start, stop or to restart the optimization process. The job progress and LS-TaSC engine output information is also displayed in this panel.

- **View:** The *view* panel is similar to a postprocessor. Various topology histories plots such as change in mass fraction, element fraction, , constraint values etc over the iterations are displayed in this panel. The *model plots* options are used to view d3plot data of iterations.

Example Problems

1. Simple linear static analysis

Objective: To set up and optimize a simple structure using LS-TaSC.

Input: A simple solid box with static loading as shown below (*bc50_extr_2s.k*)
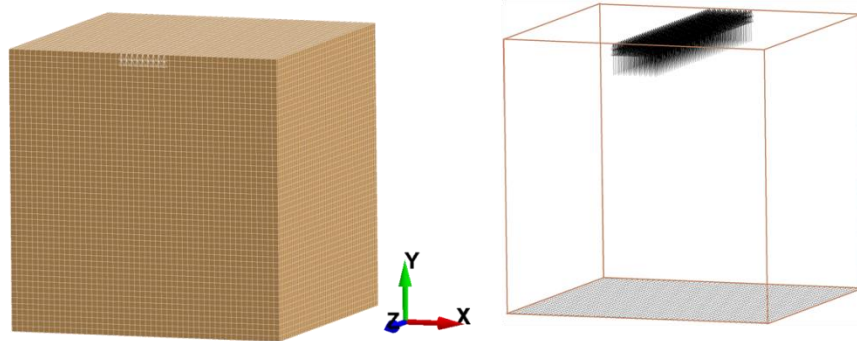
Mass fraction: 0.3
Geometric definitions: None
Constraints: None
Max. No of iterations: 30

Optimization:

➔ Open LS-TaSC 3.0 application and using *Tasc →Case →New Case* tool from right hand side of the LS-TaSC window, assign a name to the case (eg. *TOPLOAD)* and specify LS-DYNA keyword file '*bc50_extr_2s.k* as input with path of LS-DYNA executable as execution command.
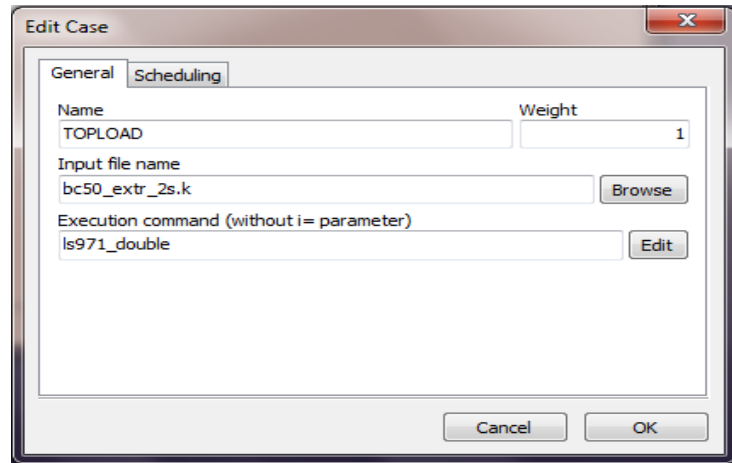
➜ Now using *Part →New*, select the part ID to be optimized and specify mass fraction as 0.3 and default values can be used for remaining fields such as minimum variable fraction and neighbor radius. No geometric definitions have been defined in this example.
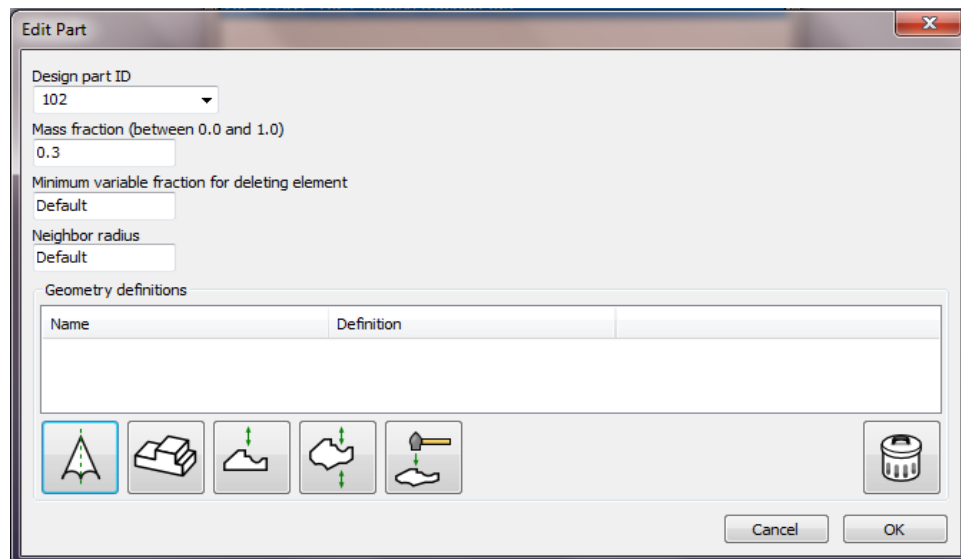
➜ Since no constraints are defined in this problem, the next step is to assign the convergence tolerance for process completion. Using *Accept* option, specify *30* as the maximum number of iterations and let the value of convergence tolerance be *Auto*.
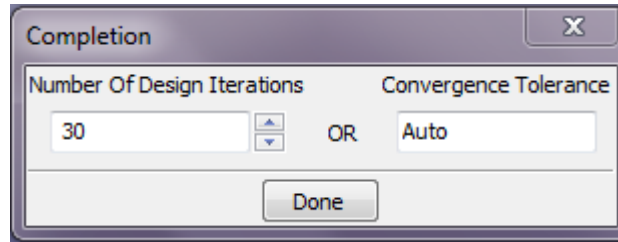
➔ Initiate the optimization process using *Run*. The *Run* window also displays the status of all jobs/iterations. The status of implicit jobs is updated when each job has been completed whereas for explicit jobs, percentage completion over time is provided.
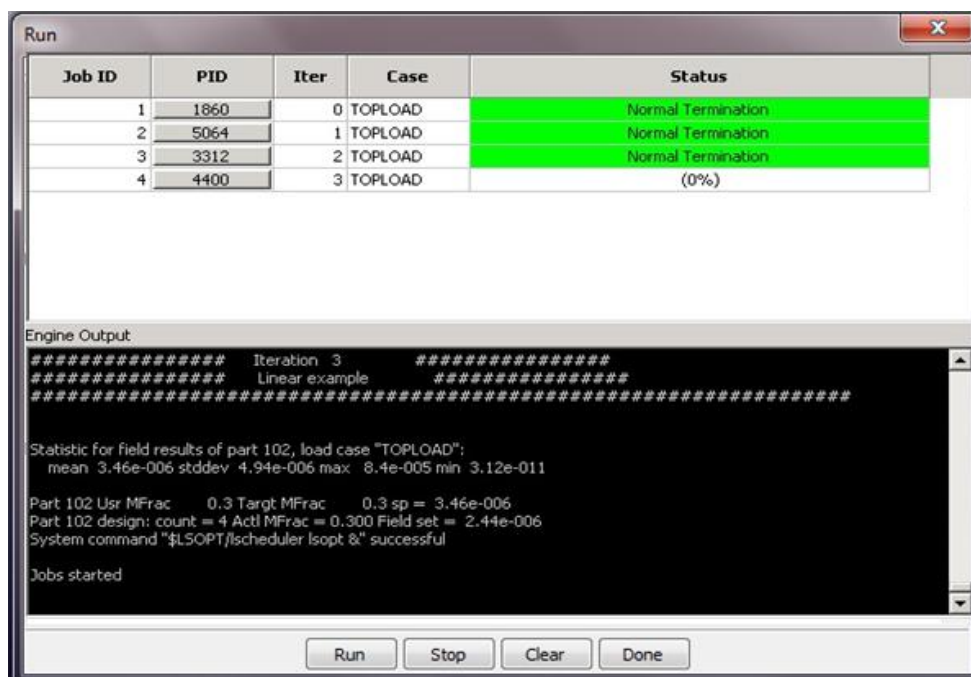
Output:

➔ The optimization process stops when convergence tolerance is met or when maximum number of iteration is reached. The output will be a new topology with reduced mass. The optimum geometry is written to a separate keyword file *OptDesign21.k* ('*21*' indicates the iteration at which the optimization process has completed).
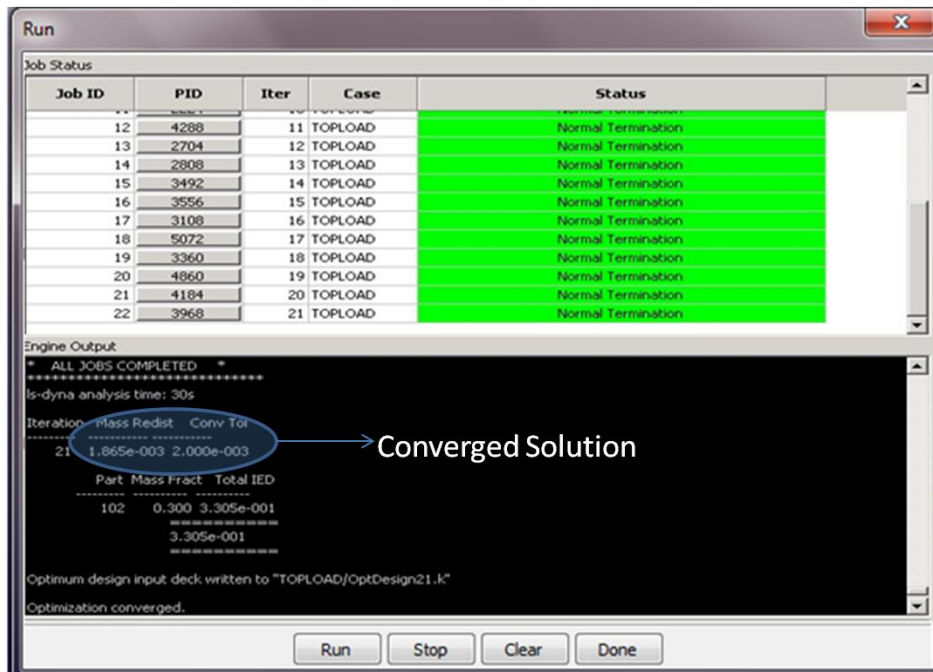
➔ Following figure shows variation of mass redistribution over optimization iterations. It can be seen that the solution reached convergence tolerance of 0.002 after 20 iterations.
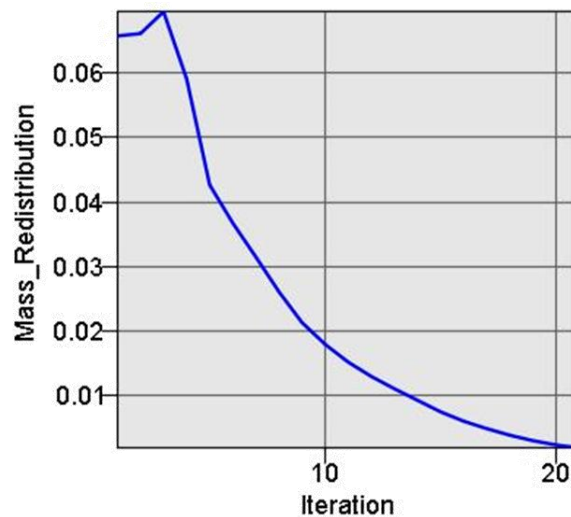
➔ Following figure is a multi plot showing the change in element fraction and mass fraction over the optimization process. There is no change in mass fraction as global design constraints were not defined in this example. The element fraction plot shows the element deletion rate through iterations. The element fraction and mass fraction plots coincide when all the elements in an iteration are full.
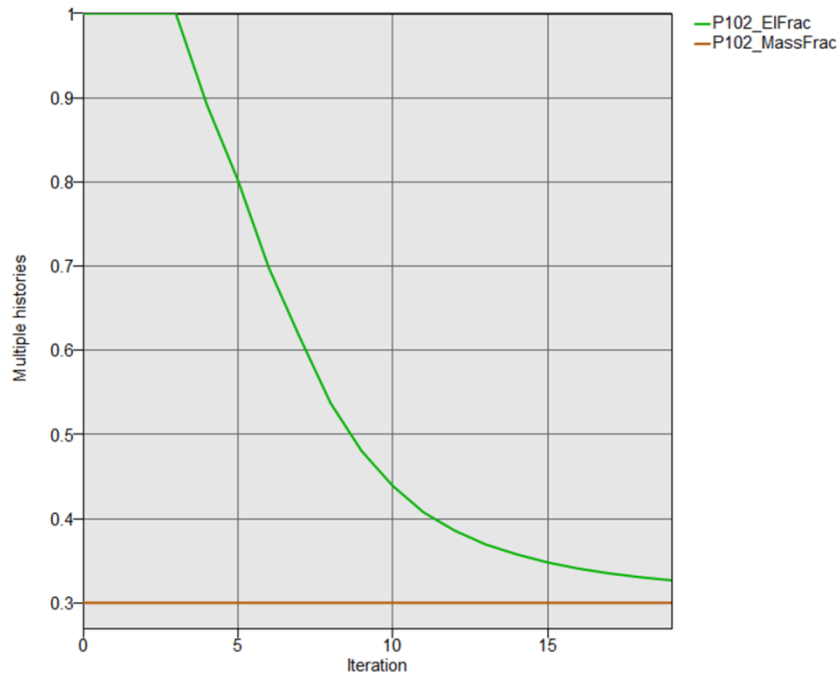
➔ The variation in topology in terms of element density over optimization process is shown below. The topology evolved by eliminating low density elements which were not contributing towards supporting the applied static loads. The region shown in red has full elements with relative density value (*x*) of 1. The final structure has wider supports at the bottom were the structure was fixed.
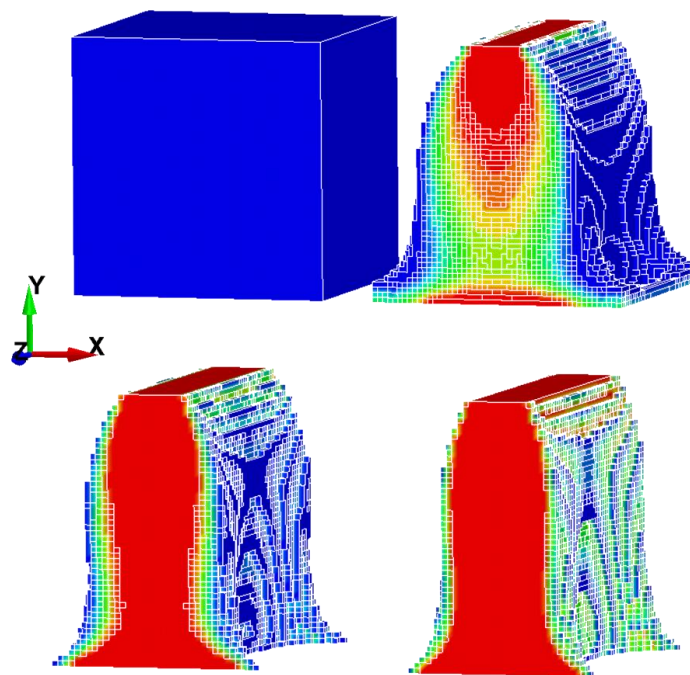
2. Optimization with goemetric definitions

Objective: To set up and optimize a simple structure using LS-TaSC with casting definition.

Input: A simple solid box with static loading as shown below (*bc50_extr_2s.k*)
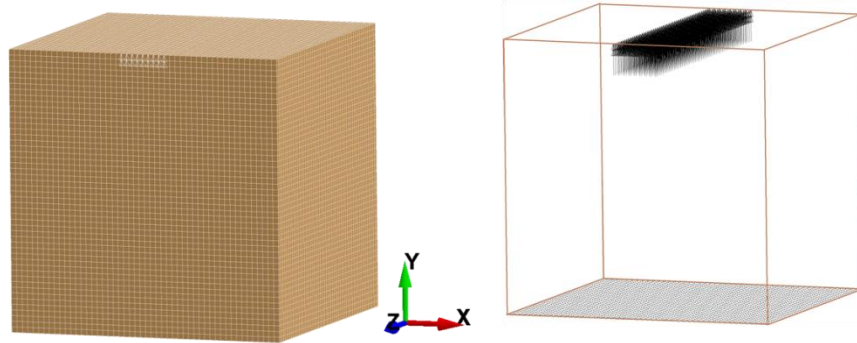
Mass fraction: 0.3
Geometric definitions: Two-way casting along Z-direction
Constraints: None
Max. No of iterations: 30

Optimization:

➔ Follow first step of previous example to set up the case dialogue box.
➔ The next step is to assign part and optimization parameters. Select part ID and assign optimization parameters similar to previous example. Using geometric definitions option, define two-way (double sided) casting along global Z direction as shown in figure below.
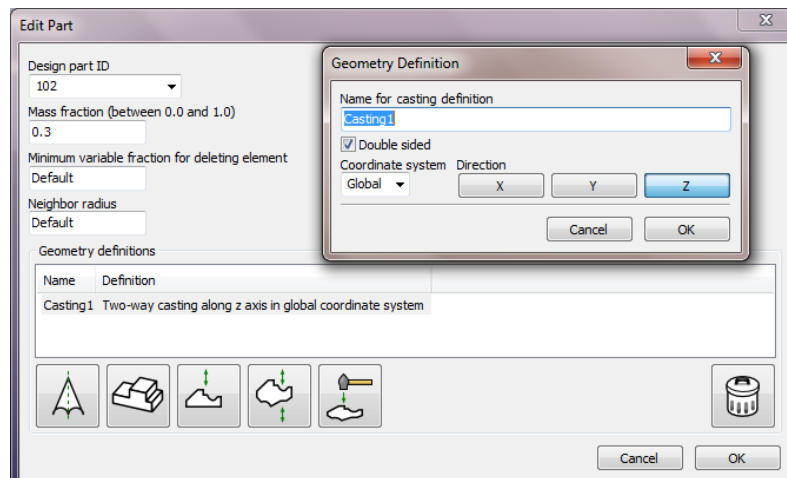
➔ The next steps include accepting the analysis job with 30 iterations and running the analysis. The solution process takes 20 iterations to converge. The following figure shows the baseline and optimum geometry obtained using two way casting definition along Z direction.
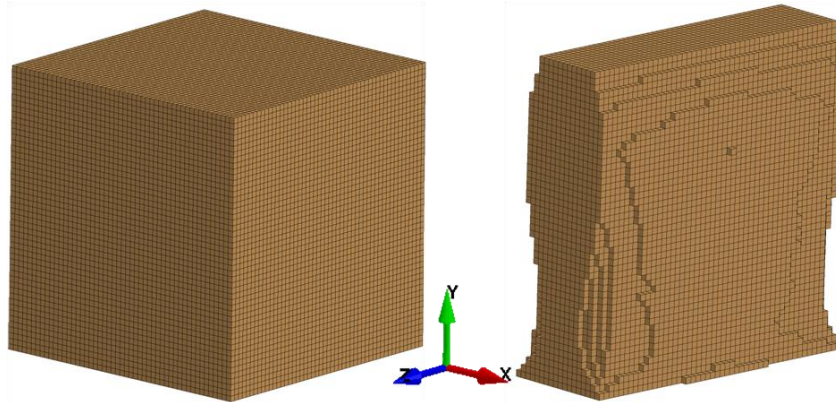

➔ The above example problem was modified by introducing symmetry geometry definition. The problem was formulated to obtain symmetric geometry along global ZX plane at middle of the structure. To do this, a coordinate system was defined as shown in figure below and plane YZ of this coordinate system was selected as the symmetric plane.
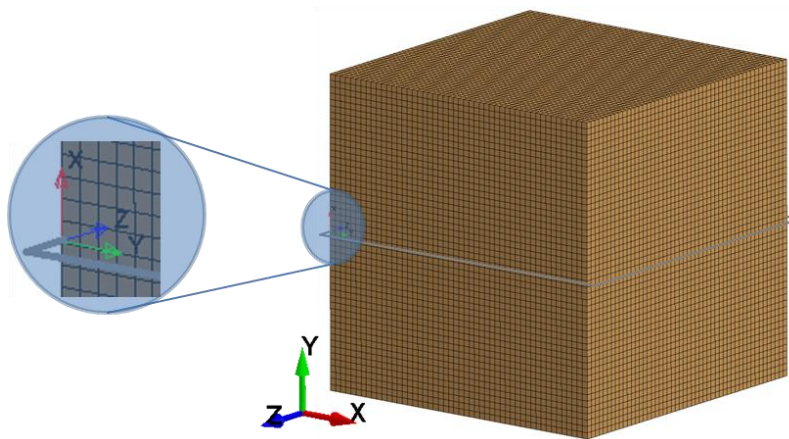

➔ The analysis setup is similar to previous example with only difference in addition of symmetry geometry definition about coordinate system *1* along YZ plane as shown in figure below.

Note: The casting definitions in LS-TaSC should be along the plane defined for symmetric definitions.
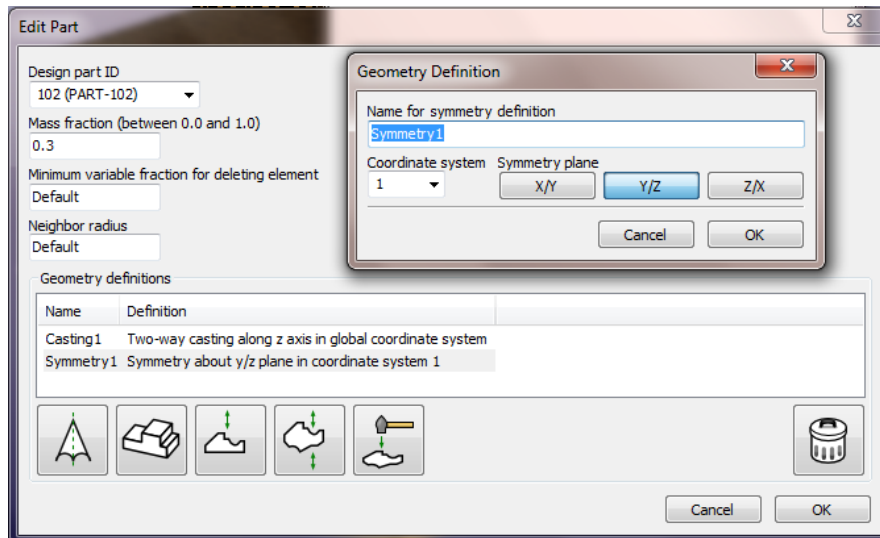

➔ The optimization process completed at 30 iterations with optimum geometry shown in figure below. It can be observed that the resulting geometry was symmetric about YZ plane of coordinate system *1*. The analysis solution did not converge but stopped as the maximum limit of 30 iterations was reached.
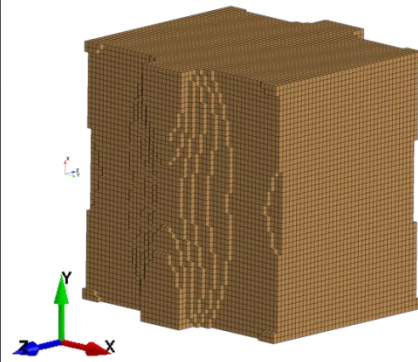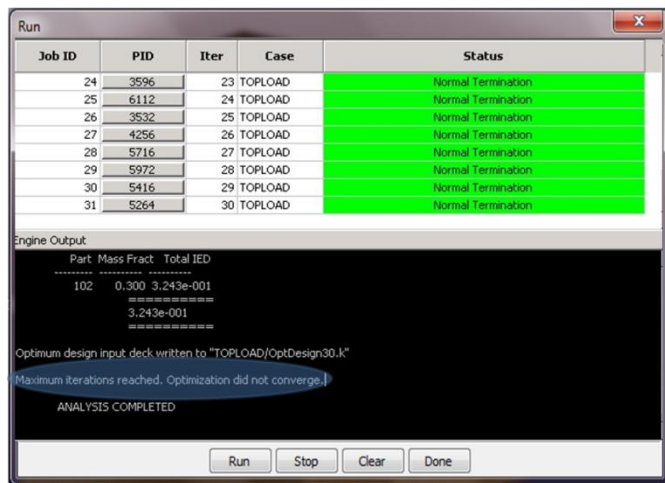
3. Optimization of a beam with global constraints

Objective: To optimize a solid beam structure under impact with global responses as design constraints.

Input: The baseline structure is a channel a beam structure with solid elements. The structure is fixed at both ends and a pole is allowed to impact at the center of the beam with a certain velocity. The beam structure is provided through LS-DYNA keyword file, *beam_LC1.k* (as shown in figure below).
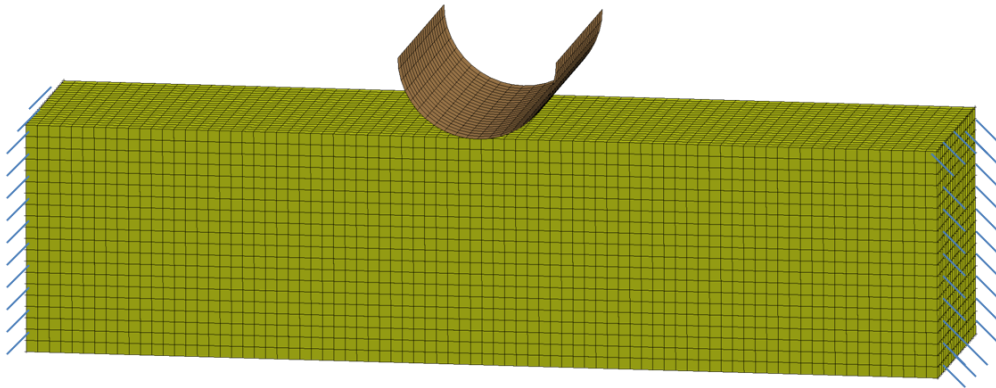
Mass fraction: 0.2
Geometric definitions: None
Constraints: Displacement, reaction force and Internal energy
Max. No of iterations: 100

Optimization

➔ Define a *Case (Clamped)* using the *Case* tool and select the appropriate LS-DYNA input file and solver executable.

➔ Using *Part* tool, select part ID 101 as design part and specify mass fraction as 0.2 with default values for other optimization parameters. Geometric definitions are not required in this example.

➔ Using *Constraints* tool, define the required three constraints. The displacement constraint can be defined using *NODOUT* constraint type by specify the node id and displacement component. Constraint bounds can be specified at options located at bottom of the constraint dialogue box.

Reaction force constraint can be defined using *RCFORC* constraint type and for internal energy, *USERDEFINED* constraint type can be used with a suitable response command (LS-TaSC accepts response commands obtained from LS-OPT). The following figure shows the *USERDEFINED* constraint dialogue box.

Note: Constraints are case specific, hence for multiple load case problems, respective case should be selected for each constraint.
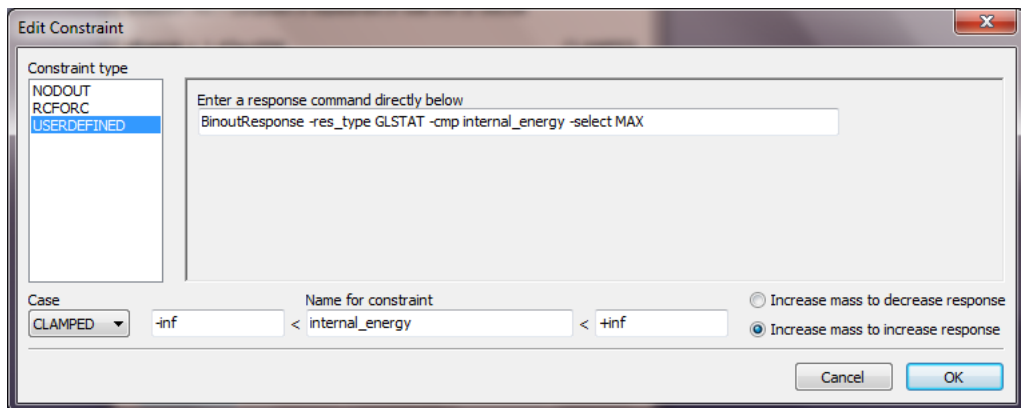
➔ *Accept* the analysis using 100 iterations as stopping criteria and using *Run* tool, start the optimization process. The solution converges at 35 iterations and the optimum geometry is shown below.
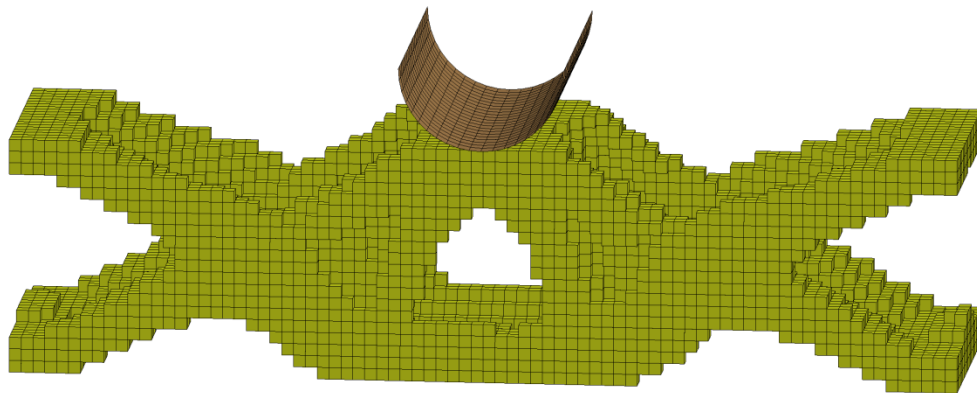

➔ The following figure shows element density contours over topology evolution.
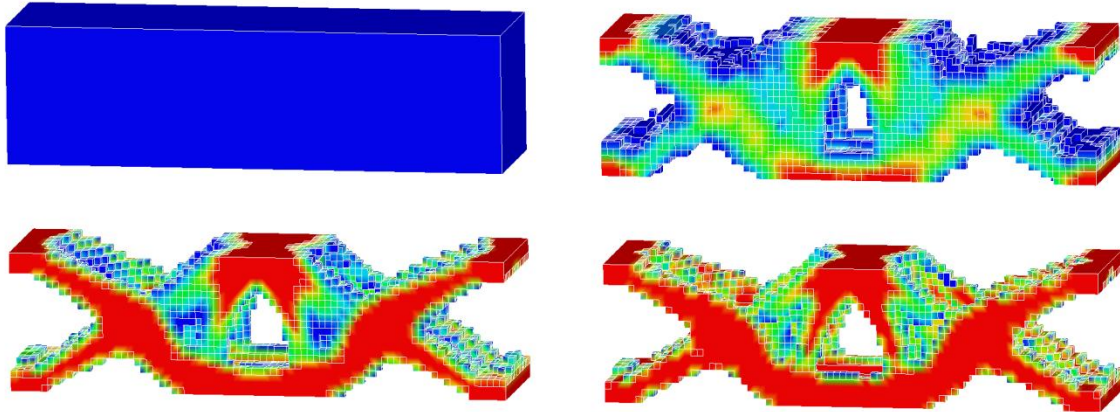
4. Topology optimization using shell elements.

Objective: Optimize a simple structure made of shell elements using LS-TaSC.

Input: The baseline structure is a channel with C cross-section provided though LS-DYNA keyword file, *sbox.k* (as shown in figure below). LS-DYNA implicit analysis is used with one side of the channel fixed and load applied along Y direction on one node on the other end of the channel.
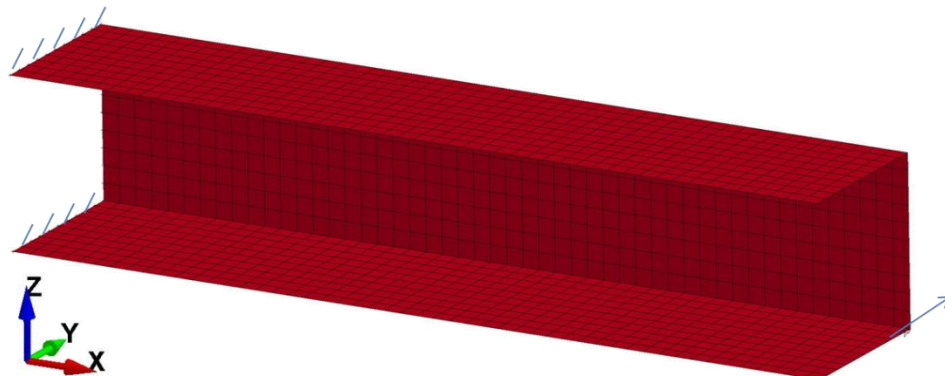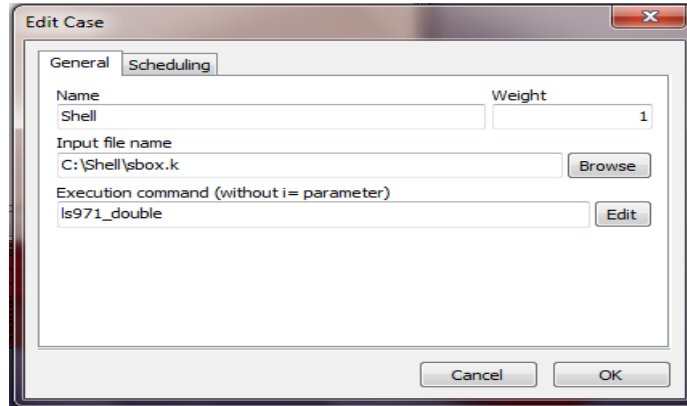

Mass fraction: 0.3
Geometric definitions: Two-way casting along Z-direction
Constraints: External work and resultant displacement
Max. No of iterations: 30

Optimization

➔ Define a case *Shell* using the LS-DYNA input file (as shown in figure below)

➔ Using *Part* tool, select the part to be optimized and specify the optimization parameters such as mass fraction etc. Geometric definitions can be defined if required.

➔ Using *Constraints* tool specify external work and displacement constraint. The displacement constraints can be defined by selecting *Nodeout* as constraint type and assigning resultant displacement for node 561 as constraint. External work response can be defined using *UserDefined* constraint type with a suitable command. The following figure shows the constraint definitions.
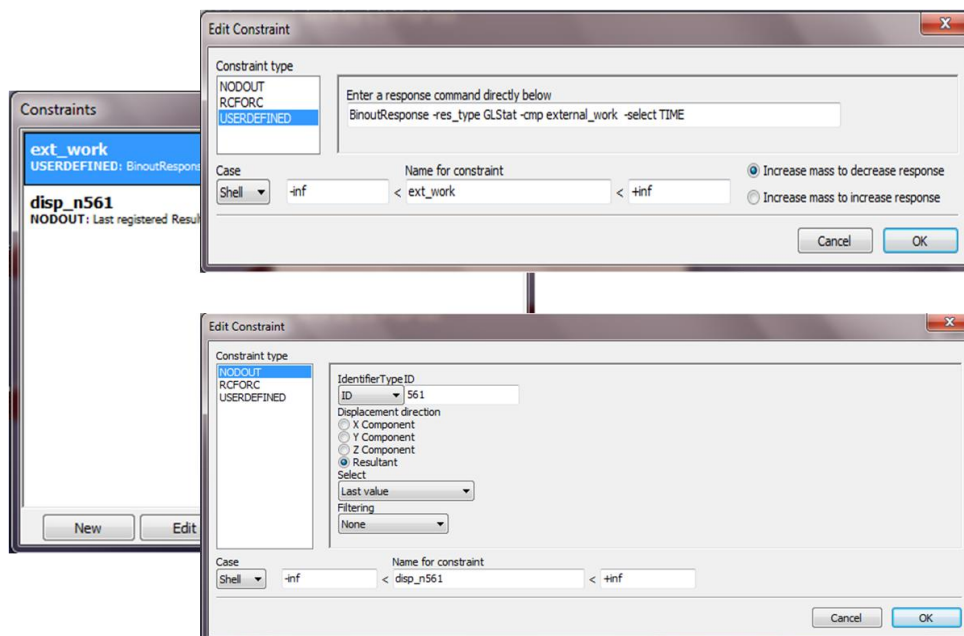
➔ Next step is to specify 30 iterations as the stopping criteria along with 0.01 as convergence tolerance for the analysis using *Accept* tool.
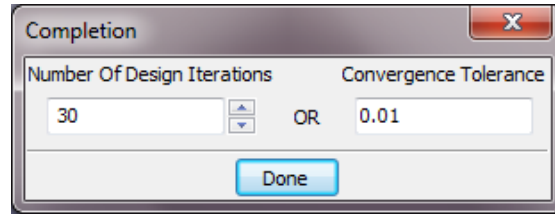
➔ The optimization process takes a total of 13 iterations to obtain a converged solution. Following figures shows the mass redistribution plot (convergence tolerance) with respect to iterations and the optimum geometry obtained in final iteration.