

Development of a Unit Cell Model for Structural Metal Sheets using

*CONSTRAINED_MULTIPLE_GLOBAL

Matthias Hartmann¹, Stephan Jäger¹, Martin Roschitz¹, Amir Horr¹

¹AIT Austrian Institute of Technology, LKR Leichtmetallkompetenzzentrum Ranshofen GmbH
Postfach 26, 5282 Ranshofen, Austria

1 Abstract

In this paper a thin structural metal sheet is researched in regard to crashworthiness as energy absorber and a *Finite Element* model based on the theory of *Representative Volume Element* is generated. Special attention is spent to the global constraining with the LS-DYNA keyword ***CONSTRAINED_MULTIPLE_GLOBAL**. It relates single displacement components of any degree of freedom (DoF). Therefore, an academic example is shown and the numerical analysis is explained. The example is restricted to the theory of linear *Finite Element Method*, containing the formulation based on the variation of displacements, the weak form of differential equations (which leads to the *Principle of Virtual Work*) and the weighting and shape functions defined by Bubnov-Galerkin. Furthermore, the benefit of reducing the system of equations and the speed-up of computation time based on the constraining of DoFs is pointed out and demonstrated by the present structural metal sheet. In addition, the functionality of a self-written python routine is exemplified, which simplifies the procedure of constraining for large three-dimensional problem domains.

2 Introduction

In *Finite Element (FE)* simulations the calculation time increases with the number of degrees of freedom (DoFs). Therefore it is aspired to keep the number of DoFs low and the system of equations small. For that purpose, one way to minimize the number of DoFs is to take advantage of symmetries. However, especially in non-linear crash simulations, this approach is not common because non-symmetric phenomena like buckling could arise. No such phenomena will occur in the reason of the applied structural metal sheet.

The structural metal sheet (see figure 1) is a three-dimensional metal sheet with small dimension in the direction of thickness. Nevertheless, this direction is the characteristic one, which properties has to be researched. In this direction the metal sheet offers folding mechanisms, which provide a controlled deformation of the macro-structural geometry. The recurring geometry leads to a periodic pattern of deformation, which can be used of during the process of FE modelling. In this case, the structural metal sheet is researched for energy absorption while crashing in small assembly dimension with small FE models using boundary conditions for a *Representative Volume Element (RVE)*. It is realized in LS-DYNA by the implementation of the ***CONSTRAINED_MULTIPLE_GLOBAL** [1] with a self-written python routine.

The field of application of this sheet is restricted to small dimensions in direction of compression and is proposed for the protection of components in highly dynamical crash situations.

Subsequently, a general overview about RVE and FEM is introduced, on which the problem definition by means of geometry and load condition is explained. Based on this, the application of the keyword is carried out.

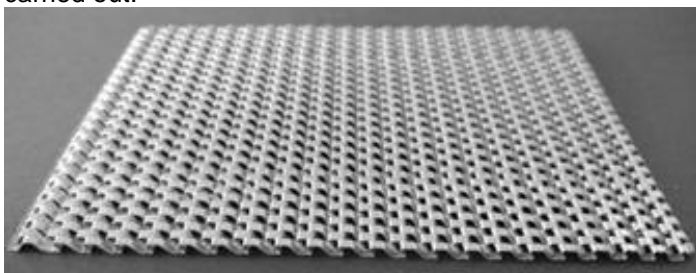


Fig.1: Structural metal sheet

3 Representative Volume Element

Structures like foams or fiber reinforced composite materials have complex and micro-structurally different material behavior. For simplified simulations and analysis this properties are often smeared in a way of continuous approach to an object with homogenous properties. Therefore, a representative volume element is necessary, as it is used in multilevel procedures like FE² method [2]. This method considers the micro scale in a way that the local properties are determined in an additional rule. The micro-structural material can be separated into micro-structure with periodicity and micro-structure with stochastically random distribution [3]. First one conforms to the structural metal sheet. The periodicity of the geometry is determined by the process of manufacturing. The only difference is the scale. The geometry of the metal sheet is not a micro-structure but a macro-structure, which has to be characterized.

Subsequently, the idea of unit cell with periodic boundaries is shown as an equivalent in the field of computational fluid dynamics. Next to the difference of Lagrangian mechanics in structural mechanics and the Eulerian mechanics in fluid dynamics, the basic mathematics can differ, but the intention of reducing the system of equations is the same. In figure 2 is shown a representative volume element of an Eulerian system with periodic boundary conditions in the direction of motion. For the structural metal sheet the Lagrangian one as the typically point of view for solid mechanics is used.

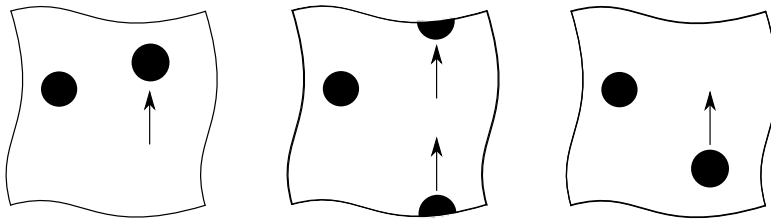


Fig.2: Unit cell with periodic boundary conditions

4 System of Equations of Finite Element Method

For further explanations of globally constrained boundaries the basic FEM is introduced on linear static behavior. Therefore, the knowledge of the method of weighted residuals, continuous Galerkin method, Bubnov-Galerkin method and discretization is introduced in a nutshell based on Hutton (2004) and Hughes (2000). For detailed information about the FEM in its entirety it is referred to the corresponding literature.

The FEM for solving boundary value problems is based on the method of weighted residuals. The residual itself in general results from a differential equation. As some complex differential equations cannot be solved with an analytical approach for any geometry, the system of equations is approximated and the error integrated over the whole domain is zero. Furthermore, the residual is weighted by an arbitrary weighting function $w_i(x)$ [4] in one dimension over the domain x as

$$\int_a^b w_i(x)R(x)dx = 0 \quad i = 1, n \quad (1)$$

The strong form of equations for a three-dimensional problem in linear static structural mechanics is given in general as

$$\sigma_{ij,j} + f_i = 0 \quad \text{in } \Omega \quad (2)$$

$$u_i = g_i \quad \text{on } \Gamma_{g_i} \quad (3)$$

$$\sigma_{ij}n_j = h_i \quad \text{on } \Gamma_{h_i} \quad (4)$$

with the Cauchy-stress tensor σ_{ij} , the body force vector f_i acting in the domain, the prescribed boundary displacements g_i and prescribed boundary tractions h_i . Now instead of x in the three-dimensional case Ω presents the problem domain and Γ is defined as the boundary of the domain

Ω , which is separated in the one with prescribed tractions Γ_{h_i} and the one with prescribed displacements Γ_{g_i} .

While equation (3) has to be fully confirmed in a strong manner, the weighted residual of equation (2) and (4) for a displacement based approach results with the Gauss' divergence theorem applied to the term $\int_{\Omega} w_i \sigma_{ij,j} d\Omega$ in the weak formulation as

$$\int_{\Omega} w_{i,j} \sigma_{ij} d\Omega = \int_{\Omega} w_i f_i d\Omega + \sum_{i=1}^n \left(\int_{\Gamma_{h_i}} w_i h_i d\Gamma \right) \quad (5)$$

Equation (5) is also known as the *principle of virtual work (PVW)*.

The Cauchy stress tensor σ_{ij} is related with the constitutive equation to the engineering strains ε_{kl} by $\sigma_{ij} = C_{ijkl} \varepsilon_{kl}$.

Furthermore, the engineering strains are derived from the displacements u_i by $\varepsilon_{ij} = 1/2(u_{i,j} + u_{j,i})$.

As the strain tensor is symmetric, it can also be transformed into a vector $\varepsilon_i = L_{ij} u_j$, with L_{ij} as the operator of spatial derivatives.

After discretizing the displacement field u_i , the traction h_i and the virtual displacements w_i with the Bubnov-Galerkin method, which means, that the weighting functions are the same as the trial functions (index h represents the discretization)

$$\begin{aligned} u_i^h &= \sum_{A \in \eta - \eta_{g_i}} N_A d_{iA} \\ g_i^h &= \sum_{A \in \eta_{g_i}} N_A g_{iA} \quad (\text{no sum on } i) \\ w_i^h &= \sum_{A \in \eta - \eta_{g_i}} N_A c_{iA} \end{aligned} \quad (7)$$

the discretized form can be written in tensor notation as

$$\underline{c}^T \int_{\Omega} \underline{N}^T \underline{L}^T \underline{C} \underline{L} \underline{N} d\Omega \underline{d} = \underline{c}^T \int_{\Omega} \underline{N}^T \underline{f} d\Omega + \sum_{i=1}^n \left(\underline{c}^T \int_{\Gamma_{h_i}} \underline{N}^T \underline{h} d\Gamma \right), \quad (8)$$

whereas \underline{c}^T can be identified as the vector of virtual displacements. Transforming equation (8) in a common way and adding the prescribed displacements to the right hand side, it can be solved and assembled to the global system of equations

$$\underline{K} \underline{D} - \underline{F} = 0. \quad (9)$$

\underline{K} is the global stiffness matrix, \underline{D} the global unknown discretized displacements and \underline{F} the global right hand side vector. The global matrix and vectors represent the whole problem domain, whereas the local matrix and vectors are

$$\begin{aligned} \underline{k} &= \int_{\Omega} \underline{N}^T \underline{L}^T \underline{C} \underline{L} \underline{N} d\Omega \\ \underline{f} &= \int_{\Omega} \underline{N}^T \underline{f} d\Omega + \sum_{i=1}^n \left(\int_{\Gamma_{h_i}} \underline{N}^T \underline{h} d\Gamma \right) \end{aligned} \quad (10)$$

Equation (9), which can also be written as $F_{int} - F_{ext} = 0$ has to be solved for \underline{D} in every FE calculation. The derivation of equation (9) gets more comprehensive for the general Galerkin method as well as for dynamic and non-linear calculations. A basic example for the simple case of linear statics is shown below for a one-dimensional problem.

5 Constrained Multiple Global

The system of equations of a recurring geometry can be minimized by suitable constraining. The most common simplifications are based on expected strains and stresses. Therefore, plane strains, plane stresses and axisymmetric stresses can be applied. Furthermore, advantage of symmetry planes can be taken. For the structural metal sheet these simplifications are not possible but a RVE can be generated and constrained in a proper way as described in the following.

The subsequent approach is for simplification restricted to the introduction of FE in the previous chapter.

5.1 Basic Idea

The keyword `*CONSTRAINED_MULTIPLE_GLOBAL` is available in LS-DYNA since version R7.0.0. It relates the degrees of freedom between nodes and corresponding direction's coordinate. An additional multiplication parameter $\alpha = COEFF$ scales the related displacements per DoF $u_{DIR}^{(NID)}$. In the LS-DYNA Keyword Manual the equation is given by LSTC (2013) as

$$\sum_{NID} \alpha_{NID} u_{DIR}^{(NID)} = 0. \quad (11)$$

Additionally to the relation between two DoFs these could be referenced to a further DoF of a third node.

In the input file the keyword is used as follows

```
*CONSTRAINED_MULTIPLE_GLOBAL
$#      ID
      101
$#      NMP
      2 (3)
$#      NID      DIR      COEF
      1          1        1.0
      5          1       -1.0
      (NodeREF   1        1.0) Reference
```

The number of constrained multi points (`NMP`) has to be adapted dependent on the existence of a reference node from two to three. The possibilities of constraining in this keyword are restricted to the Cartesian coordinates in x, y and z-direction. Several dependencies are realized by the combination of mixing directions and coefficients. Thereby angle dependencies etc. can be defined by the user. A basic example with constrained nodes based on the keyword definition above without reference node, a concentrated force on node 5 and a mounted node 3 is shown in figure 3a). The example is carried out in detail in the following chapter. There are also the results depending on the input parameters discussed.

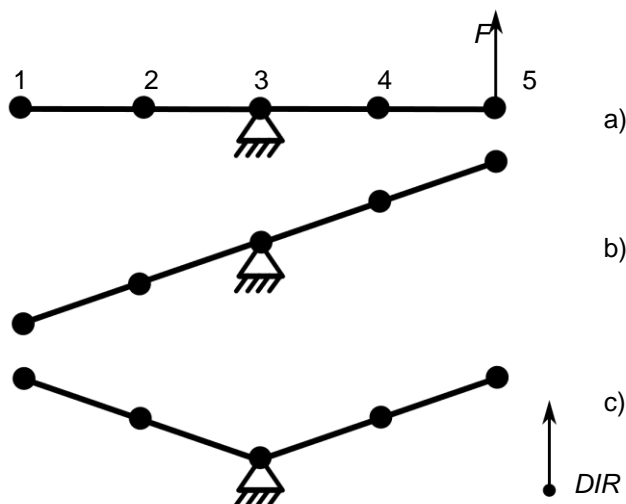


Fig.3: 5 DoFs system with transverse displacements

5.2 Constrained Degree of Freedom in System of Equations

Next to the geometry and the number of nodes, the DoFs in the global system of equations can be minimized with `*CONSTRAINED_MULTIPLE_GLOBAL`, too. Therefore, the effects of the keyword based on the FE-theory are explained subsequently in a simple way building on the FE theory explained in chapter 3. For a simple bar system (see figure 3) with DoFs restricted to the transverse direction and a single node load on node 5, the global matrix notation for five nodes and correspondingly five DoFs is

$$\begin{bmatrix} K_{11} & K_{12} & \cdots & 0 \\ K_{21} & K_{22} & K_{23} & \ddots & \vdots \\ & K_{32} & K_{33} & K_{34} & \\ \vdots & \ddots & K_{43} & K_{44} & K_{45} \\ 0 & \cdots & & K_{54} & K_{55} \end{bmatrix} \begin{bmatrix} D_1 \\ D_2 \\ D_3 \\ D_4 \\ D_5 \end{bmatrix} - \begin{bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \\ F_5 \end{bmatrix} = 0, \quad (12)$$

because there is just one direction $DIR=1$, $u_{DIR}^{(NID)} = u_1^{(NID)} = D_{NID}$ follows. The direction index is neglected for further proceeding. Using the transformed equation (11) for constraining D_1 and D_5 , $D_1 = -\alpha_1 / \alpha_5 * D_5 = \beta * D_5$ results. Inserting in equation (12), the system of equations changes to

$$\begin{bmatrix} K_{11} & K_{12} & \cdots & 0 \\ K_{21} & K_{22} & K_{23} & \ddots & \vdots \\ & K_{32} & K_{33} & K_{34} & \\ \vdots & \ddots & K_{43} & K_{44} & K_{45} \\ 0 & \cdots & & K_{54} & K_{55} \end{bmatrix} \begin{bmatrix} \beta * D_5 \\ D_2 \\ D_3 \\ D_4 \\ D_5 \end{bmatrix} - \begin{bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \\ F_5 \end{bmatrix} = 0. \quad (13)$$

This can be transformed to a smaller system of equations

$$\begin{bmatrix} K_{22} & K_{23} & 0 & \beta * K_{21} \\ K_{32} & K_{33} & K_{34} & 0 \\ 0 & K_{43} & K_{44} & K_{45} \\ 0 & 0 & K_{54} & K_{55} \end{bmatrix} \begin{bmatrix} D_2 \\ D_3 \\ D_4 \\ D_5 \end{bmatrix} - \begin{bmatrix} F_2 \\ F_3 \\ F_4 \\ F_5 + F_1 \end{bmatrix} = 0. \quad (14)$$

In consideration of the assumption of displacement based elements, the displacement D_1 is constrained and has not to be tested by a weighting function. Thus, the first column and the first row can be omitted.

$$k_{ij} = \begin{bmatrix} k_{11} & k_{12} \\ k_{21} & k_{22} \end{bmatrix} = \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \quad (15)$$

If the example in figure 3 is applied with an elemental stiffness matrix k_{ij} as in equation (15) and a single nodal force $F_5 = 1$, the following system of equations (14) and the scaling coefficients $\alpha_1 = 1$, $\alpha_2 = -1$, $\beta = 1$ has to be solved. Row three and column three can also be neglected, because $D_3 = 0$ is already known by the fixed boundary condition (see figure 3a)). The system of equations is reduced to

$$\begin{bmatrix} 2 & 0 & -1 \\ 0 & 2 & -1 \\ 0 & -1 & 1 \end{bmatrix} \begin{bmatrix} D_2 \\ D_4 \\ D_5 \end{bmatrix} - \begin{bmatrix} 0 \\ 0 \\ 1 \end{bmatrix} = 0, \quad (16)$$

which results in

$$\begin{bmatrix} D_2 \\ D_4 \\ D_5 \end{bmatrix} = \begin{bmatrix} 1 \\ 1 \\ 2 \end{bmatrix}, \quad D_1 = D_5 = 2$$

The result is shown in figure 3b). The evaluation of $D_1 = D_5$ is done during the post processing. For choosing the scaling parameters as $\alpha_1 = 1$, $\alpha_2 = 1$, figure 3c) ensues. The constrained nodes 1 and 5 are moving in opposite directions.

5.3 Application to Structural Metal Sheet

In a first step the structural metal sheet is investigated with regard to its compression behavior and energy absorption. With the intent of defining a homogeneous material characterized by the behavior under compression, a unit cell based on the idea of a RVE is modelled. Considering chapter 3, it follows as shown in figure 4. The nodes on the front edges of the dark gray elements are the constrained boundary nodes, which are related in a manner, that the original metal sheet is represented as in figure 1. Thus, the front side and the backside as well as the left and the right side are coupled by `*CONSTRAINED_MULTIPLE_GLOBAL`.

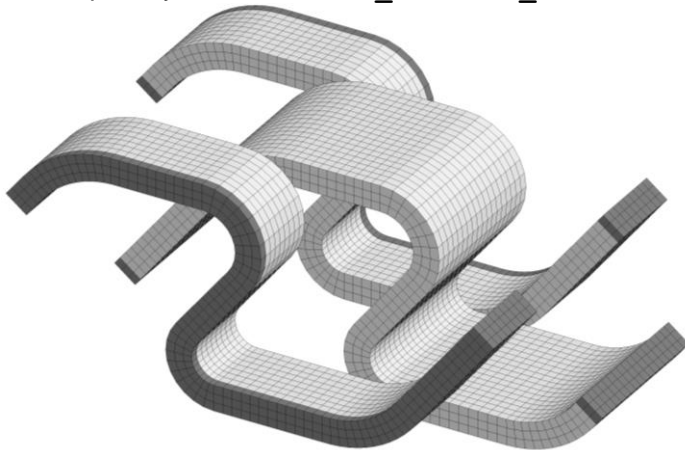


Fig.4: RVE of the structured metal sheet

In contrast to the previously discussed example in figure 3, it is recommended to constrain either, the translational and rotational freedoms of the dark gray elements. To model this behavior, the available element formulations are restricted to solid elements, because the rotational DoFs of shells cannot be coupled. Next to the contact issue between the single rows of structural metal sheet, this is a reason for using solid elements, too. In figure 5 the compressed sample is shown. There, the rotated ends are visible.

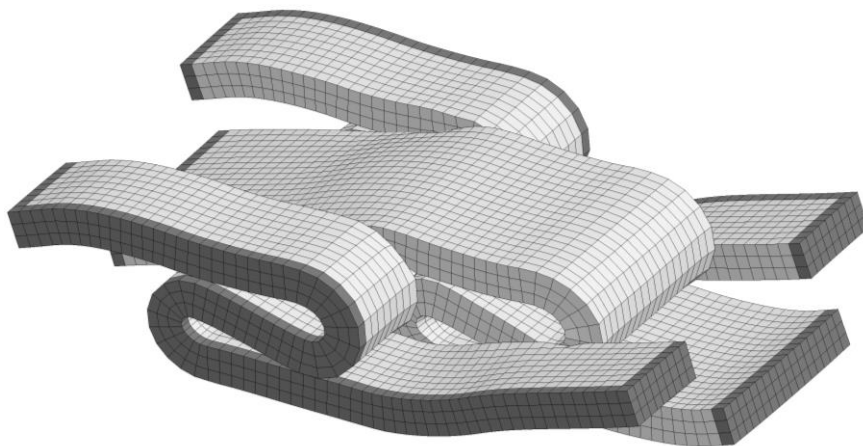


Fig.5: compressed RVE of the structured metal sheet

5.4 Results of Global Constraining

To return to the topic of reducing the DoFs, in chapter 5.2 one DoF is eliminated by global constraining. For the approach of structural metal sheet 1560 pairs of DoFs are constrained by a total number of 46548 DoFs. This means, that by an increasing number of constrained degrees of freedom, the size of the system of equations decreases. Thus, the system of equations and also the calculation time can be reduced dramatically. The speed-up itself depends on the numerical solver. To check the number of DoFs and the size of system of equations to solve, in LS-DYNA the optional card `*CONTROL_IMPLICIT_SOLVER` for implicit solving brings some more information about the system of equations. The `LPRINT` can be added for additional output about statistics on model-oriented memory and cpu requirements [1]. Furthermore, with `MTXDMP` the system matrix and vectors as in equation (9) are written in an output file [1].

6 Python Routine for Automatic Constraining

For models with a huge number of nodes and DoFs, there is a lot of manual constraining because the LS-PrePost does not support the constraining of `*CONSTRAINED_MULTIPLE_GLOBAL` in a proper way. In order to create the input file with the corresponding keyword every node and its constrained node has to be related. There is no possibility of picking elements by surfaces or any other geometry. The remedy is an automatic constraining based on geometrical parameters of the FE model. Therefore, a Python routine was programmed, which reads in the nodes and elements from the `.k`-file, generates the constrained node pairs and additionally writes out a LS-DYNA input file. For applying the routine in a proper manner, the FE model has to be prepared. The routine constrains nodes, which are part of different surfaces. In addition for the treated model, the nodes are constrained by their accordance of Cartesian coordinates in two dimensions. In which manner (means direction and coefficients) the surfaces or nodes are constrained is defined by input parameters through the command line. An additional user defined deviation parameter respects numerical deviations in direction of coordinates. To speed up the constraining by the routine, the number of nodes in a constrained partID has to be reduced.

A small example is shown in figure 6. The cube consists of *five* parts: the gray one in the middle is not used for constraining. The two light gray parts contain the nodes, which have to be constrained. These nodes are also part of the dark gray top and bottom shells. The input for the command line comprises the partIDs, which have to be constrained. In this case, the nodes of the dark gray shells are constrained. The light gray solids are not used for this example, but they can replace the shell elements. Additionally, the direction of constraining has to be entered, which is for an overall constraining 1,2,3 for x-,y- and z-direction. Thereupon, the routine checks all nodes of the input partIDs considering a deviation defined by the user. The deviation can be based on the geometry or numerical deviations of coordinates.

Obviously this routine is restricted to special geometries and meshes, but the self-written routine is highly adoptable, because of the module based scripting.

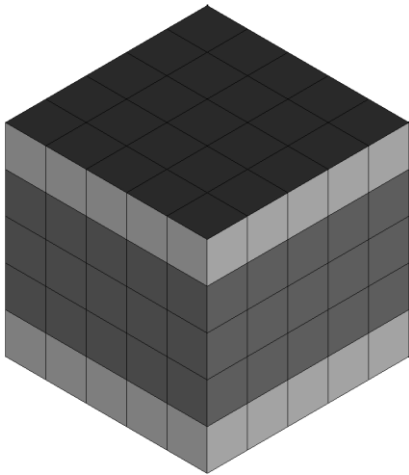


Fig.6: Constraining with Python Routine

7 Summary

A structural metal sheet is investigated regarding energy absorption under impact loading. For getting information about the behavior, a unit cell model based on the idea of a RVE is modelled. This serves as the basis for a homogeneous material model, which could be developed in the future. The modelling of the RVE is provided by the recurring geometry and pattern of deforming while compression. Thus, the whole metal sheet can be reduced to a smaller model and globally constrained DoFs. This is realized by the new keyword ***CONSTRAINED_MULTIPLE_GLOBAL**. Based on the FE theory some restrictions are set for further proceeding. The restrictions for a small one-dimensional example are the displacement based approach, a continuous Galerkin method, especially the Bubnov-Galerkin method with equivalent shape and weighting functions and the simplification of linear statics. For the dynamically computed structural metal sheet, these restrictions are not applicable, but the basic idea of globally constraining of the system equations is equal. Since the elements are based on a continuous Galerkin method, this procedure would have to be the same.

Next to the theory, the keyword is applied to the structural metal sheet and the advantage of eliminating DoFs is shown. It results in a smaller system of equations and a decreasing computation time. Additionally, for a fast and efficient constraining a python script is written, which generates a *.k*-file for input in LS-DYNA. The algorithm utilizes the details about the recurring geometry of the treated metal sheet. For that reason, the routine does not satisfy on any geometry, problem domain and mesh, but the algorithm can be modified in regard to the special problem.

8 Literature

- [1] LIVERMORE SOFTWARE TECHNOLOGY CORPORATION (LSTC): "LS-DYNA® KEYWORD USER'S MANUAL VOLUME I", 2013, Version R7.0, p.10-102, 10-103; pp. 12-204 – 12-209
- [2] Zienkiewicz, O.C.; Taylor, R.L.: "The Finite Element Method for Solid and Structural Mechanics", 6th Edition, 2005, pp. 567-572
- [3] Li, S.; Wang, G.: "Introduction to Micromechanics and Nanomechanics", 1st Edition, 2008, p. 73
- [4] Hutton, D.V.: "Fundamentals of Finite Element Analysis", 1st Edition, 2004, pp. 131-132
- [5] Hughes, T.J.R.: "The Finite Element Method: Linear Static and Dynamic Finite Element Analysis", 1st Edition, 2000, pp. 75-92