

Modeling Net Capture of an Object in LS-DYNA

Lee-Hee Drory, Aviner Shreiber, Matan Elbaz, Omer Livneh

Rafael Advanced Defense Systems

1 Abstract

One way of capturing a moving object is by using a hanged cage-like net. The process is to be reliable under various impact velocities and orientations of the captured body. Thus, accurately modeling its dynamics is vital due to the need to account for multiple probable outcomes while including the complex net structure. The focus of the present work is on the modeling, characterization and calibration of the net and its motion. To best represent the geometric properties of the net, its initial state, motion, and body capturing procedure, the following stages were carried out. First, a beam-element-based model of the net was constructed in MATLAB® using a generative function, that allows simple generation of a multiple-parameter dependent net structure. This was due to the need for flexibility in dimensions, geometric properties, part allocation and base unit cell shape. Second, mechanical properties of the hyper-elastic unique net structure were characterized by a series of tensile experiments followed by properly choosing LS-DYNA elements and material formulations. Finally, two sets of finite element analyses (FEA) were conducted, where the first included folding the net to determine its initial state for the successive impact simulation. Both folding and impact procedures required careful consideration and examination of different aspects, such as contact types and formulations, mechanical forces and damping.

2 Net Structure

This chapter reviews the properties and parameters of the actual net structure, as later needed for creating the computational model.

2.1 Geometry

The net consists of the following general components, as shown in Figure 2.1:

- A) Five-sheet cage-like net structure made of Dyneema® twines.
- B) Double net layer impact zone.
- C) Steel guide cables, fixed on both ends.
- D) Steel hooks designated to allow the net to slide along the cables.
- E) Kevlar straps connecting the net to the hooks.

In addition to the base net structure, several other features were modeled as indicated in Figure 2.2. The outlines of each sheet are 25mm wide Dyneema® straps, designed to frame the sheets and the openings. Additionally, 12mm wide diagonal Kevlar straps are placed in the bottom and top sheets for strengthening.

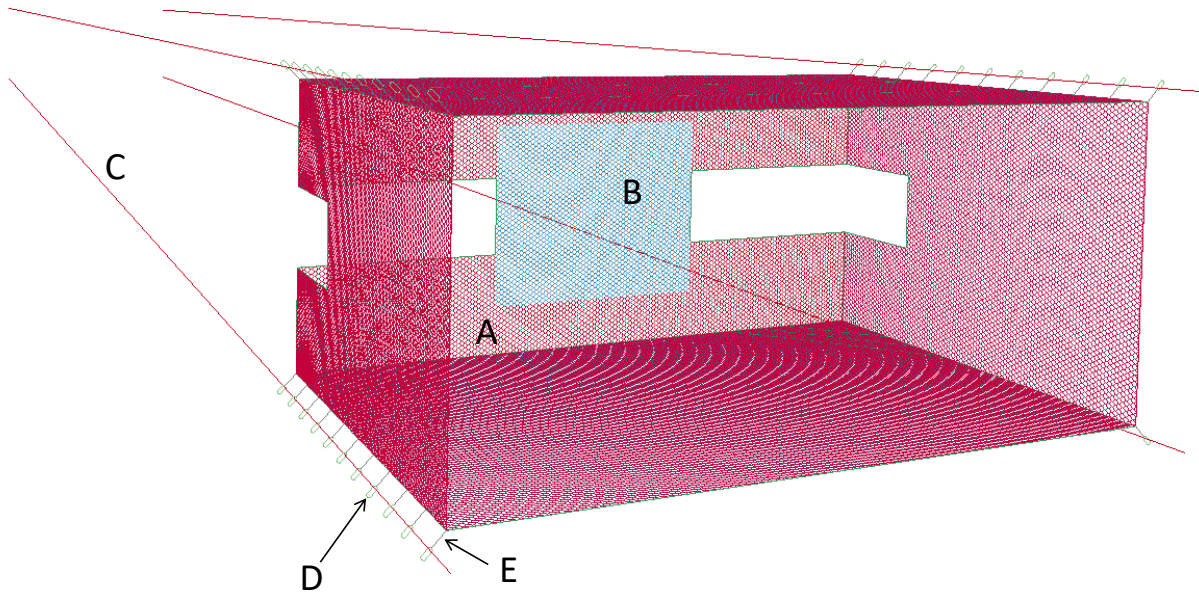


Figure 2.1: Base net structure components

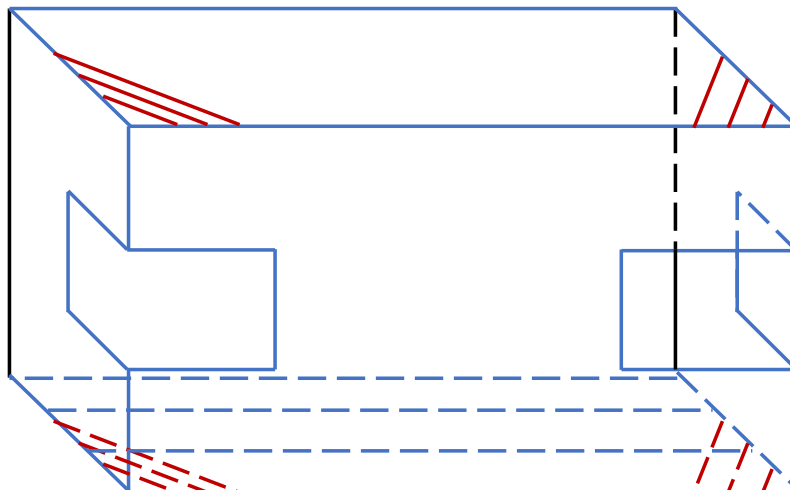


Figure 2.2: Strengthening straps: Dyneema frame straps (blue) and Kevlar strengthening straps (red).

2.2 Unit Cell and Twine Features

The base structure of the net is made of 5 perpendicular faces, comprised of twines tied to each other at constant intervals. This structure creates hexagonal unit cells, each of which consists of two components: twines and knots, as shown in Figure 2.3. In order to simulate the difference between twines and knots, they are modeled as two distinctive materials as described in chapter 5.

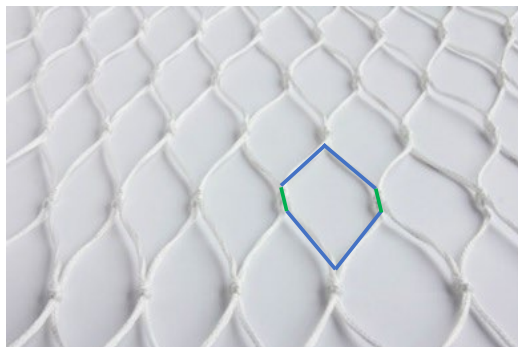


Figure 2.3: Net and unit cell

3 Generative Code

As described in chapter 2, the net consists of different components with different materials and geometries. Dimensions of each component should be allowed to vary per need, thus a highly flexible generative MATLAB[®] code of the net was built, enabling control of these dimensions, while using a few simple parameters. The code creates an output in the form of a key file, containing geometry data in the form of nodes and elements. The following subchapters detail the needed parameters and generative algorithm.

3.1 Input Parameters

The input parameters of the code were divided into two main groups:

- a) General dimensions of the net. These are listed in Table 3.1 and visualized in Figure 3.1.
- b) Dimensions of the unit cell and other components.

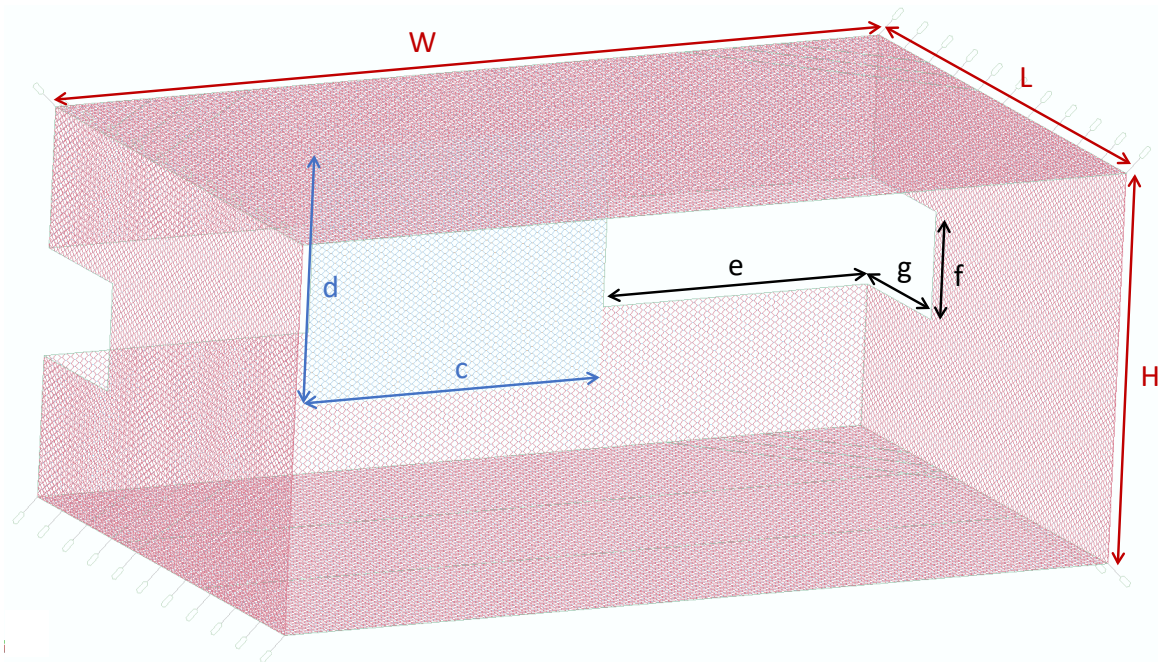


Figure 3.1: General geometrical parameters of the net.

| Parameter | Meaning | Parameter | Meaning |
|-----------|--------------|-----------|----------------|
| W | Cage width | d | Target height |
| L | Cage depth | f | Opening height |
| H | Cage height | g | Opening depth |
| c | Target width | e | Opening width |

Table 3.1: List of the net's general dimensions parameters

Parameters of the unit cell are visualized in Figure 3.2, where a is the knot size, b is the twine size, and α is the characteristic angle. Values of general height and length (h and t , respectively) are subsequently derived. Other code input parameters are detailed in Table 3.2.

| Parameter | Meaning | Parameter | Meaning |
|-----------|------------------------|-----------|---|
| a | Unit cell's knot size | k | Number of hooks |
| b | Unit cell's twine size | l_{cd} | Hook's distance from cage |
| α | Characteristic angle | n_d | Number of diagonal strengthening straps |
| h_c | Hook height | nb_{sb} | Number of bottom sheet strengthening straps |
| t_c | Hook length | g_l | Guide cables' length |

Table 3.2: List of unit cell's dimensions and related parameters.

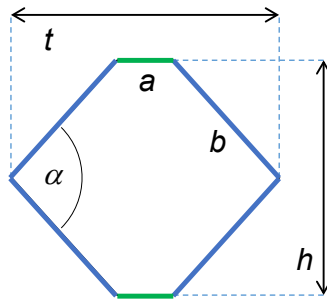


Figure 3.2: Unit cells are generated from knot and twine sizes (a, b), as well as the angle α . all other parameters are consequential.

3.2 Net Generation Scheme

The parameters mentioned in chapter 3.1 are used to generate the complete net structure. The flowchart depicted in Figure 3.3 demonstrates a general outline of the generative algorithm.

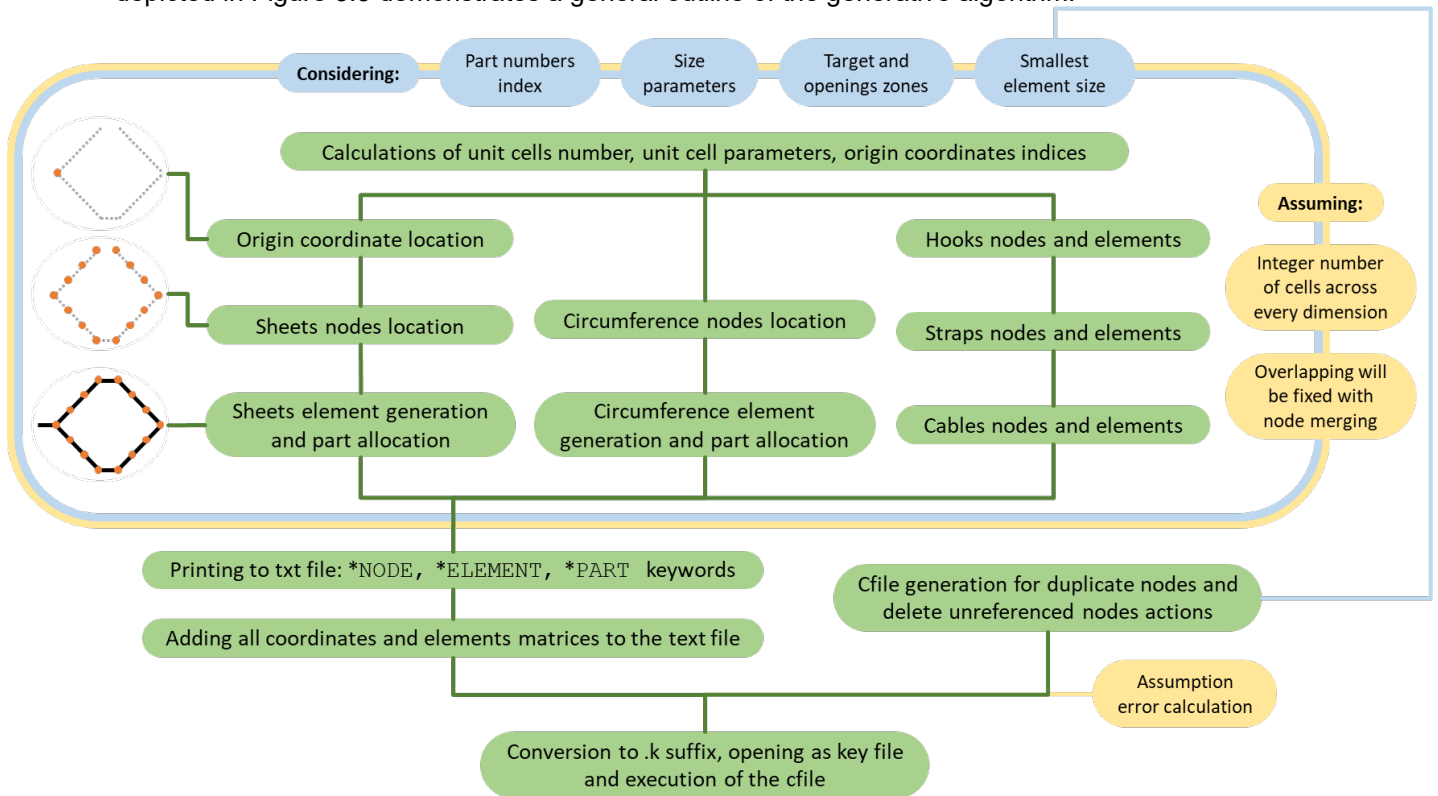


Figure 3.3: Scheme of the net generation process

3.3 Unit cell and sheet generation

The unit cell of the net is generated according to given origin coordinates (x_0, y_0) and the three main parameters a , b and α . The code generates equally spaced nodes, with space size a , so that two adjacent nodes create an element. Each unit cell contains only one knot, while the other knot is associated to the next cell, in order to prevent unnecessary duplication. The nodes and elements' coordinates can be seen in Figure 3.4.

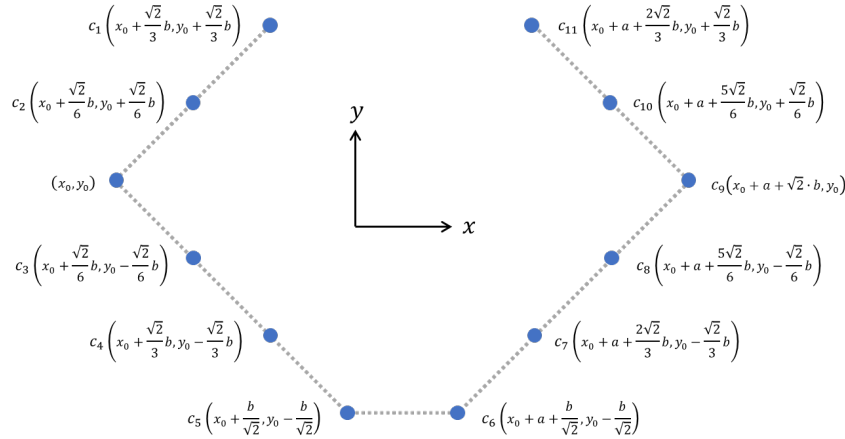


Figure 3.4: Generating a unit cell from an origin point (x_0, y_0) , based on parameters a , b and α .

4 Net Segment Characterization

The net is made of Dyneema® twines, tied at uniform intervals to create an array of twines and knots. To analyze its dynamic behavior properly, a characterization of mechanical properties was required – for both twines and knots separately. Since the knots cannot be modeled explicitly as tied twines, an equivalent property for them had to be defined.

For this reason, tensile experiments were designed and performed to determine those properties. Such an experiment should be compared to a compatible analysis, in order to characterize and refine both twine and knot equivalent material properties to get an accurate simulation of the net's actual behavior as resulted from the experiment.

4.1 Experiments

Two characterization experiments were performed as follows:

1) Single Dyneema twine tensile test

A single Dyneema twine was placed in the tensile machine and was stretched until failure. The experiment was duplicated several times, for which data of tensile force and displacement were recorded. The typical load for failure was approximately 5.5kN. A photo of the experiment setup is shown in Figure 4.1. The experiments were conducted in the Structures Under Extreme Conditions Lab, at Ben-Gurion University (BGU).



Figure 4.1: Single Dyneema twine tensile test

2) Net segment tensile test

A net segment of 10*10 cells was loaded in a tensile machine (Figure 4.2) and stretched with a displacement rate of 50mm/min. As in the single twine experiment, the displacement and force data were recorded. These experiments were held in-house, using Shimadzu tensile machine and custom designed net holders.

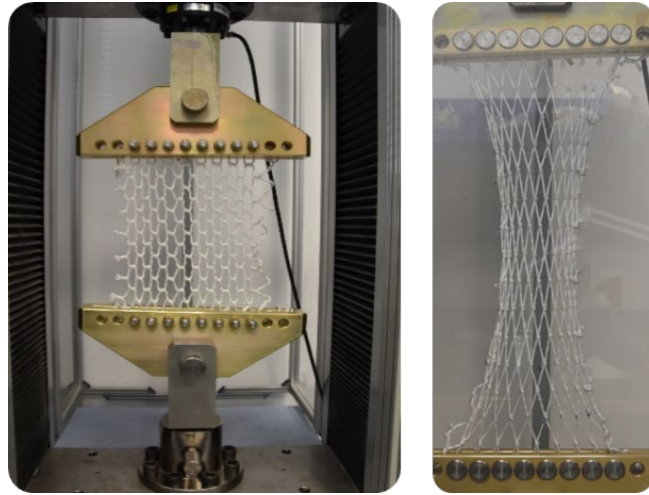


Figure 4.2: Net segment tensile test. Left: Unstretched, right: stretched.

4.2 Finite Element Analysis

A FEA was conducted to obtain equivalent material properties for the twines and knots, in order to match the behavior of the analysis to the net segment tensile test. In the analysis, several combinations of different properties between the twines and knots were inspected, resulting in choosing the one that yielded the best agreement with experimental data for the net capturing LS-DYNA simulation.

A parametric static analysis of the net segment was performed in ANSYS Mechanical (APDL solver), modeling the net as beams with different properties for the twine and knot as shown in Figure 4.3. In compliance with the experiment, the bottom loops of the segment were fixed with solid cylinders, and the top loops were assigned displacement for stretching of the net.

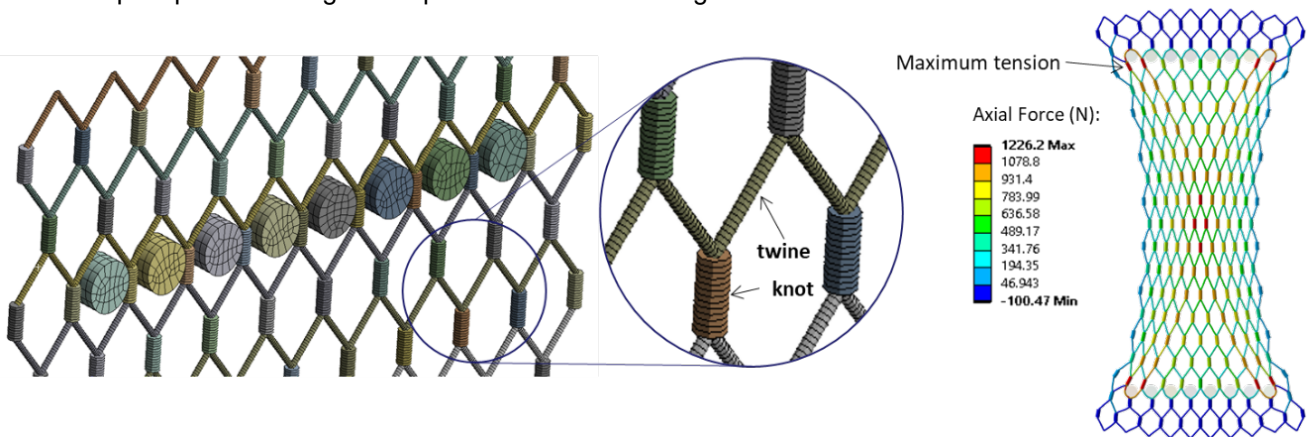


Figure 4.3: Analysis of the net segment

4.3 Results and Outputs

The results of the single twine and the net segment tensile tests are shown in Figure 4.4 (a) and (b) respectively. The single twine test results demonstrate a hyper-elastic behavior of the material. The net segment results show a less significant but similar behavior, as well as fluctuations in the tension force that can be associated with a dynamic behavior of the knots.

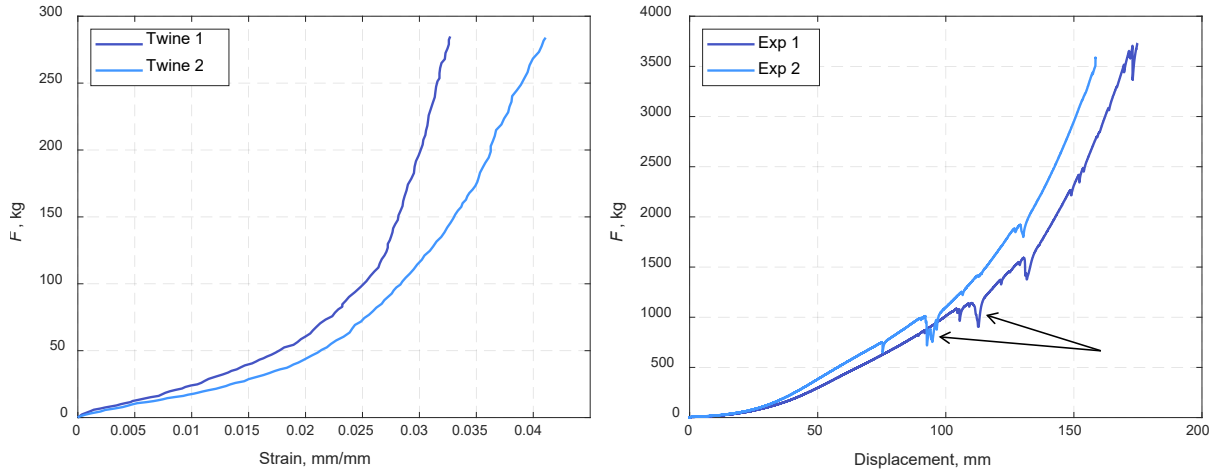


Figure 4.4: Tensile experiment results for: (a) a single twine, showing force vs. strain, and (b) net segment, showing force vs. displacement.

5 Dynamic Simulations

An LS-DYNA simulation of object capturing and net motion was conducted. The following subchapters detail the analysis setting and modeling of the folding and impact processes.

5.1 Element Formulations and Materials

All the assembly components were modeled as beams with three different element formulations (Elform) based on their functionality:

- 1) Elform=6 (discrete beam/cable): Twines, Strengthening straps, Guide cables, Kevlar straps (for hook's connection).
- 2) Elform=3 (truss): Knots.
- 3) Elform=1 (Hughed-Liu cross section integration): Hooks.

The material formulation is correlated with the element formulation. Knots and hooks are modeled as 001-MAT_ELASTIC, while all other materials are modeled as 071-MAT_CABLE_DISCRETE_BEAM.

The purpose of choosing 001-MAT_ELASTIC with Elform=3 to the knots and hooks is to simulate their full material properties geometrically. Knots and hooks can experience tension, bending and compression, and it is important to consider these properties in the model.

In contrast, 071-MAT_CABLE_DISCRETE_BEAM with Elform=6 considers only tension stiffness, and it is a more efficient choice for the strengthening straps, guide cables and Kevlar straps since they mostly experience tension during the simulation.

Twines do experience bending and compression during the folding and impact, but their stiffness in those modes are relatively small compared to tension.

5.2 Folding Simulation

The purpose of the folding process is to bring the net from the generated model to the actual net's state before the impact. The folding simulation consists of two main actions: applying gravity to the model and applying a displacement condition to the first hook, causing the net to fall and hang with the hooks on the guide cables, as well as the back part of net to move towards the front. In addition, a point in the top sheet of the net was pulled upwards to prevent the body from hitting the limp top sheet. The final state of the folding analysis is saved as a dynain file for initialization of the impact analysis. The result of this part of the simulation is a hanged and folded net as shown in Figure 5.1.

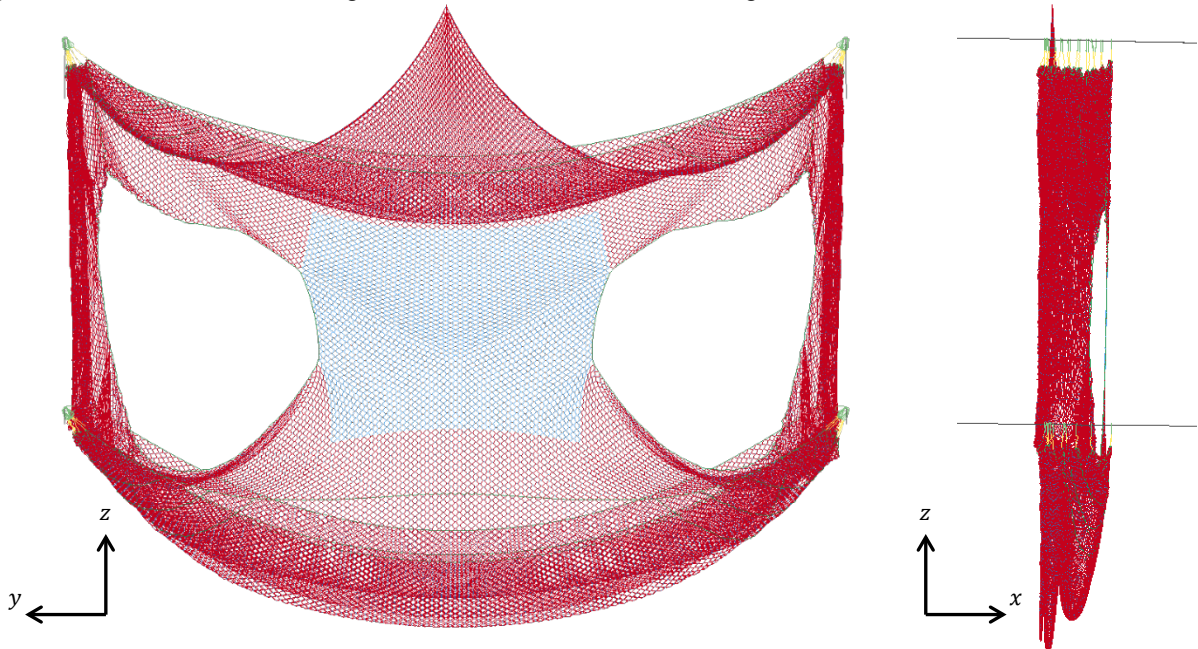


Figure 5.1: Folded net, front and side views

The boundary conditions for the folding stage were determined as follows:

- 1) Gravity was inserted via `LOAD_BODY_Z`. In order to prevent large fluctuations during the fall, a `DAMPING_GLOBAL` keyword was inserted. A decreasing linear curve of damping coefficient was set to allow the falling to be neither too fast nor too slow according to the folding progress, to both complete the drop and ensure that the net's movement is fully damped within the termination time.
- 2) Using `BOUNDARY_SPC`, the ends of the guide cables and the first four hooks (on the left side in the side view of the folded net, Figure 5.1) were fixed. While the ends of the cables were fully fixed, the hooks' movement was limited only in the x axis, preventing them from sliding along the cables but still allowing them to rest and rotate.
- 3) The method chosen for the folding is applying a displacement condition via `BOUNDARY_PRESCRIBED_MOTION_SET` on a node set containing the four last hooks. The displacement condition brings the hooks in a constant velocity up to the desired location of the hooks.

The contact keyword for the whole model was chosen to be `AUTOMATIC_GENERAL`, due to the large number of beam elements. Though we found beam elements contacts to be best represented in this manner, this contact type is considered computationally heavy resulting in a long solving time. Moreover, the analysis experienced large fluctuations within the net, hook and cables, resulting in large unphysical internal energies. To solve this issue, two parameters were altered in the contact card:

- Scale factor of the penalty stiffness for both 'surface a' and 'surface b' (SFS, SFM) was changed to 0.1 (default: 1)
- Viscous damping coefficient percent (VDC) changed to 20 (default: 0). A larger value of VDC did not offer any further improvement. A comparison between the internal energy with and without VDC can be seen in Figure 5.2.

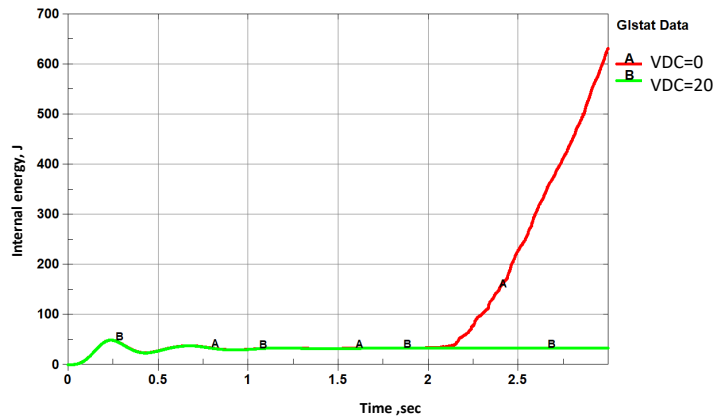


Figure 5.2: Internal energy in the folding simulation with and without VDC

5.3 Impact Simulation

Unlike the folding analysis that needs to run once for every set of net dimensions, the impact analysis includes several study cases, thus a shorter solution time is more critical. To achieve that, most of the contacts were removed, except for the ones between the hooks and cables. The contacts within the net are much less significant when the impact occurs because the net stretches out and does not fold in, so that it is assumed there is no contact between net elements. The contact between the hooks and the cables remained as `AUTOMATIC_GENERAL`, and there was no need to change SFS, SFM and VDC settings from their defaults. The contact between the rigid body and the net was set to `AUTOMATIC_NODES_TO_SURFACE`.

To slow down the body and the net after the impact, there are four brakes connected to each corner of the net via the first hook. In the model, the brakes are modeled as `ELEMENT_DISCRETE` with one node merged with the first hook, and the other end fixed to the ground. The brake elements were associated with `MAT_S05_DAMPER_NONLINEAR_VISCOUSE`, with a load curve defining a braking force correlated with the velocity of the first hooks. A demonstration of the impact motion is presented at a certain time instance in Figure 5.3.

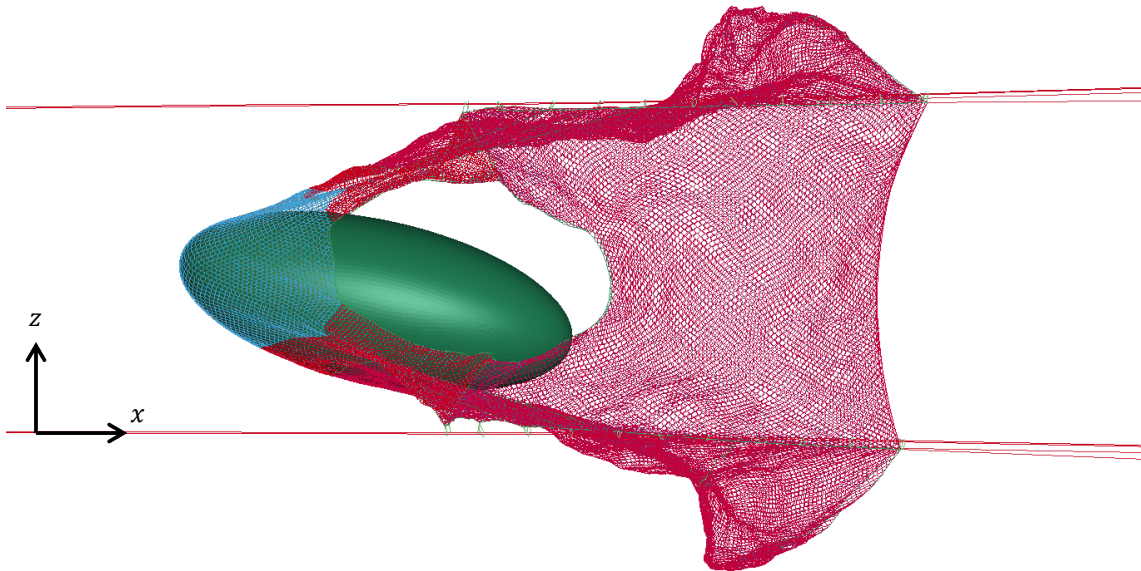


Figure 5.3: The impact analysis of a body captured by the net

6 Conclusion

This study has undertaken an exploration of modeling a complex net structure in LS-DYNA, through flexible generation in a modular code, consideration of material properties, contact definition and element formulation, as well as preliminary analyses and experimentation.

The resulted simulation represents every modeled mechanism, from sliding of the hooks along the guiding cables, motion of the net and the interaction with the captured body. This analysis enables conducting a comprehensive parametric study to assess the dynamic behavior of the capturing process in multiple scenarios and operating conditions.