UNIVERSITÀ DEGLI STUDI FIRENZE

UNIVERSITÀ DEGLI STUDI FIRENZE
DICEA
DIPARTIMENTO DI INGEGNERIA CIVILE E AMBIENTALE

Road safety through FEM simulations: concepts and criteria towards a 0-deaths strategy

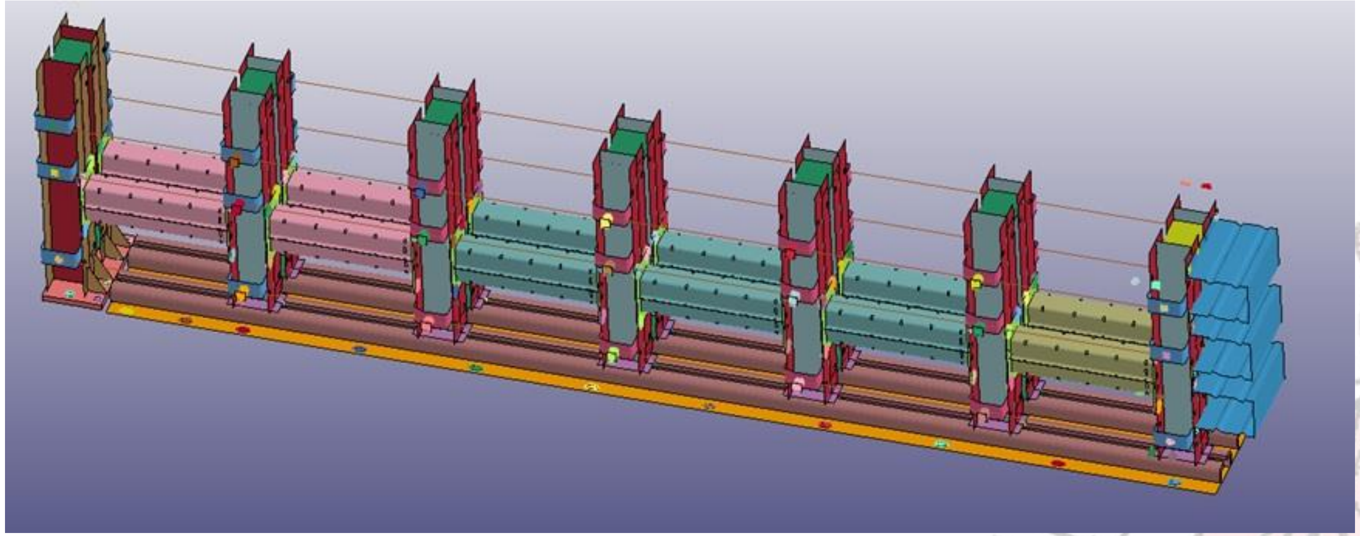
Topic Introduction

Phd. Eng. Monica Meocci

September, 09 - 2019

The FEM Methods

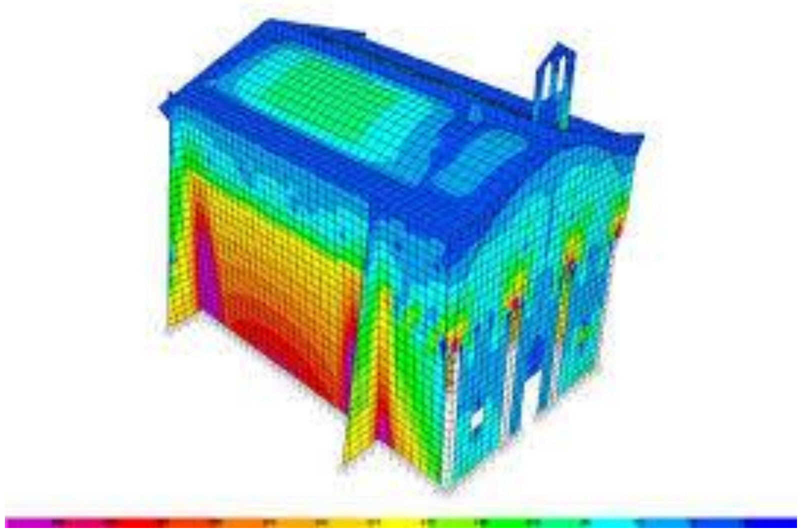
The **finite element method** (**FEM**) is a numerical method for solving problems of engineering and mathematical physics.

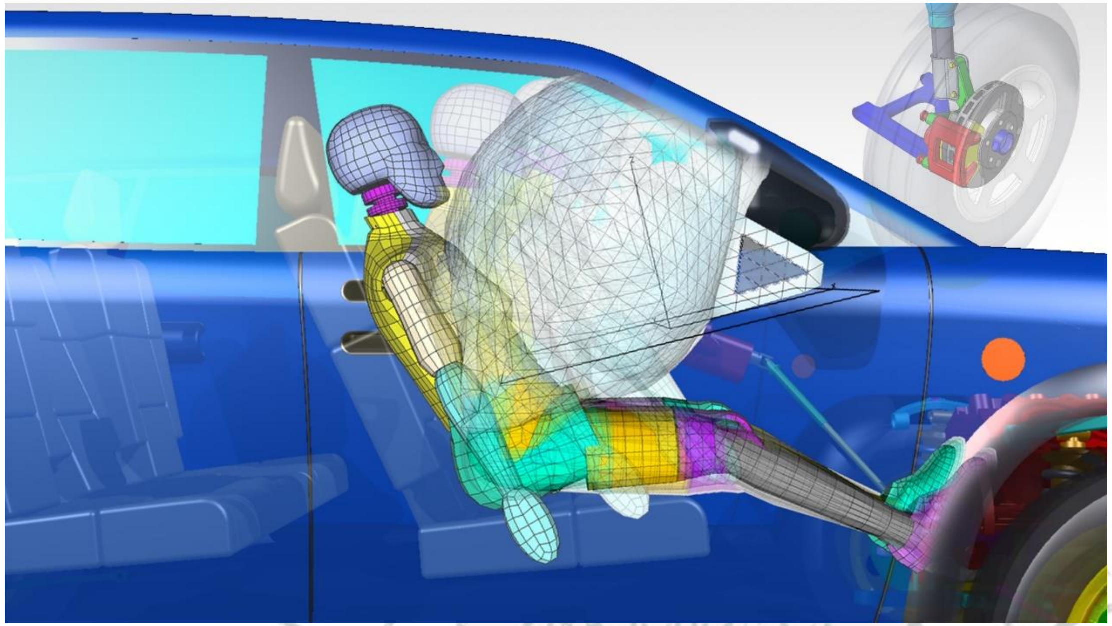


To solve the problem, it subdivides a large system into smaller, simpler parts that are called finite elements. The simple equations that model these finite elements are then assembled into a larger system of equations that models the entire problem.

The FEM Methods

The **finite element method** (**FEM**) is a numerical method for solving problems of engineering and mathematical physics.



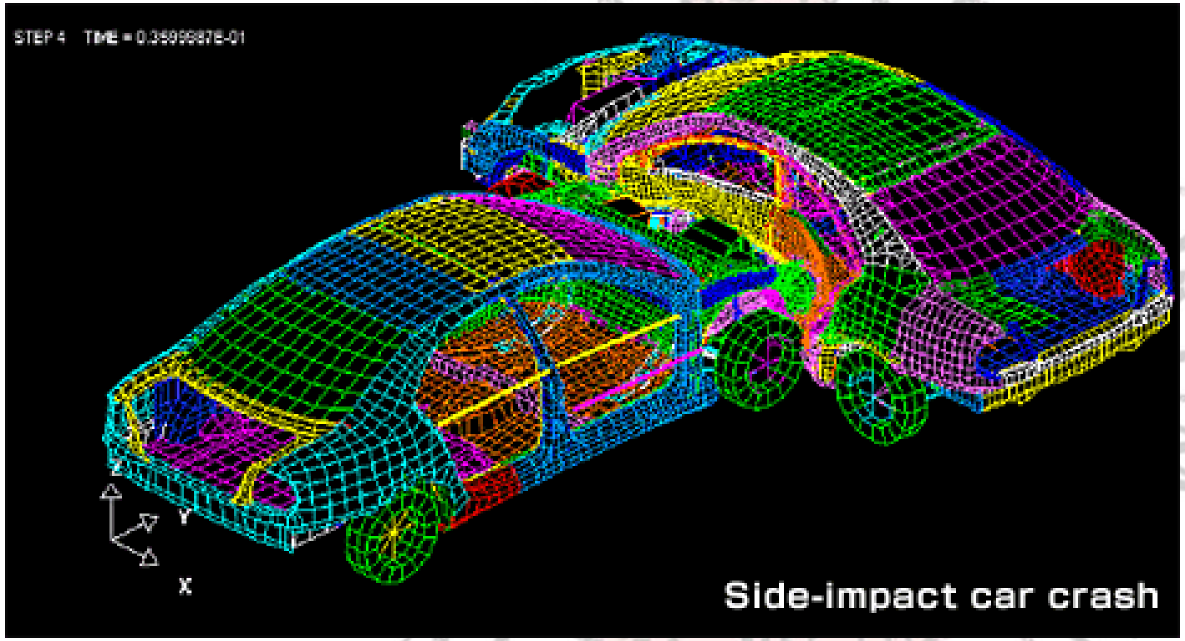


To solve the problem, it subdivides a large system into smaller, simpler parts that are called finite elements. The simple equations that model these finite elements are then assembled into a larger system of equations that models the entire problem.

LS DYNA Software

LS-DYNA is a general-purpose finite element program capable of simulating complex real world problems.

It is used by the automobile, aerospace, construction, military, manufacturing, and bioengineering industries. LS-DYNA is optimized for shared and distributed memory Unix, Linux, and Windows based, platforms, and it is fully QA'd by LSTC. The code's origins lie <u>in highly nonlinear, transient dynamic finite element analysis using explicit time integration.</u>



Nonlinear

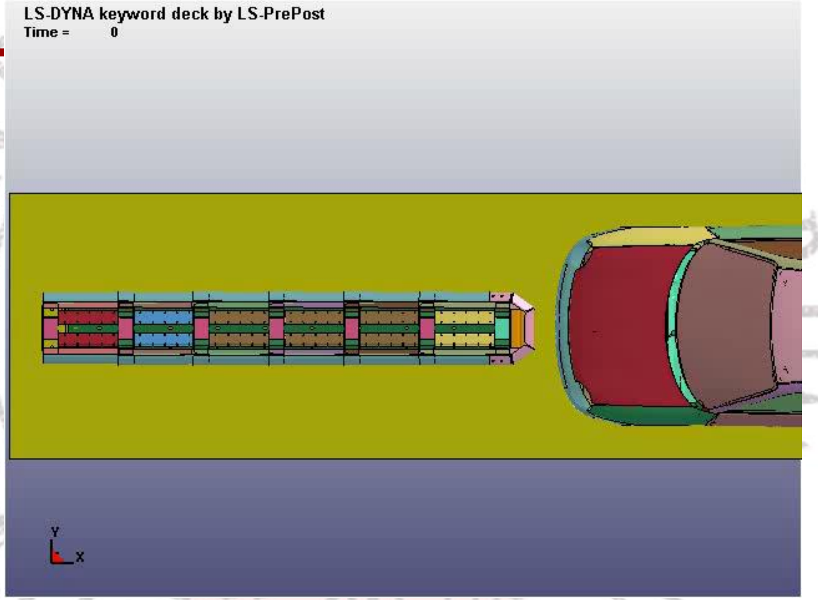
- Changing boundary conditions (such as contact between parts that changes over time);
- Large deformations (for example the crumpling of sheet metal parts);
- Nonlinear materials that do not exhibit ideally elastic behavior (for example thermoplastic polymers).

Transient dynamic

…means analyzing high speed, short duration events where inertial forces are important.

Typical uses include:

- Automotive crash (deformation of chassis, airbag inflation, seatbelt tensioning);
- Explosions (underwater Naval mine, shaped charges);
- Manufacturing (sheet metal stamping).



LS DYNA Software

Need and characteristics:

It is appropriate to investigate and solve problems characterized by:

- large deformations;
- sophisticated material models;
- complex contact conditions (with the possibility of automatically managing the contact areas); and
- working in time domain;
- modelling a wide range of material and their behaviour;
- models different types of elements.

Main issues to be consider:

- Complexity of the physical phenomenon;
- Interaction between multiple objects → contacts, connections and penetration;
- Material behaviour according to the speed of the system;
- Secondary effects due to the application of "loads" (speed, forces, forcing, etc.).

Main issues to be consider:

These conditions imply a high complexity in the evolution of the phenomenon and a very variable response of the studied system.

Added to this …the complexity of the modelling of the boundary conditions variable during the evolution of the phenomenon over time.

The system is therefore based on the resolution of a system composed of the following three classes of equations:

- Equilibrium equations;
- Compatibility equations;
- Bonding equations.

Equilibrium equations:

Equilibrium equations relate stresses to applied forces.
Hp: linear equations for small displacements

$$[M]\ddot{u}(t) + [C]\dot{u}(t) + [K]u(t) = f(t)$$

$$[K]u(t) = f(t)$$

Static analysis

Where [M], [C] and [K] are the matrix of masses, damping and elasticity respectively.
The three vectors represent velocity and acceleration displacements respectively.

Compatibility equations:

Compatibility equations relate deformations to displacements.
Small deformations → linear equations

$$\varepsilon_x = \frac{\partial u}{\partial x} \qquad \varepsilon_y = \frac{\partial v}{\partial y} \qquad \varepsilon_z = \frac{\partial w}{\partial z}$$

$$\gamma_{xy} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \qquad \gamma_{yz} = \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \qquad \gamma_{zx} = \frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}$$

→ from which the internal congruence equations are derived

If the deformation components respect the internal congruence equations, the congruence of the deformation is guaranteed

No penetration!

LS DYNA Software

Bonding equations:

The binding equations describe a constitutive empirical relationship that can be of various types…(elastic, elastic-plastic, thermal…)

$$\sigma = f(\varepsilon, \dot{\varepsilon})$$

Where ε, $\dot{\varepsilon}$ represent the deformation of the material and its velocity deformation.

LS DYNA Software

$$[M]\ddot{u}(t) + [C]\dot{u}(t) + [K]u(t) = f(t)$$

The analytical solution of the "linear" case is available in a closed form

$$[M]\ddot{u}(t) + [C]\dot{u}(t) + \boxed{[K(u)]u(t)} = f(t)$$

Of more interest is the resolution of the "non-linear" case, that is when, at each integration step, the matrices can change (being a function of time)

iterative numerical integration methods

- Implicit methods;
- Explicit methods.

→ Newmark

LS DYNA Software

explicit codes → generally based on the central differences methods.

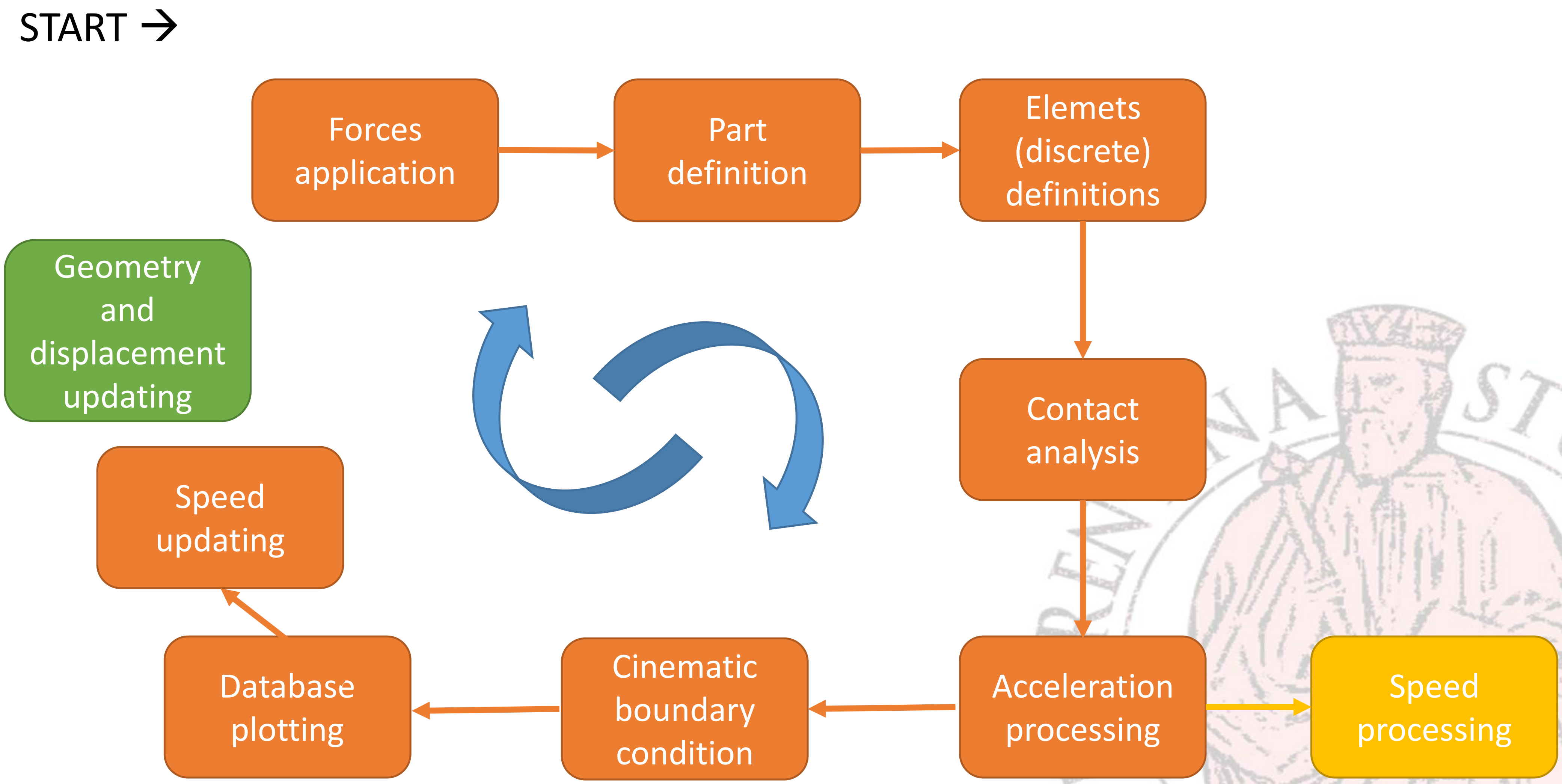
The equations of equilibrium at the nodes are written in the configuration for which both the displacement and the speed are already known, so that once the acceleration has been calculated, it is possible to proceed with integration over time.

$$u_{n+1} = u_n + \Delta t \times f(u_n, t_n)$$

The solution to a generic time does not depend on itself, but only on the solution at the previous instant.

The most used method of this type is the integration of finite differences.

LS DYNA Software



LS DYNA Software

advantages

Eliminating the problem of having to invert stiffness matrix at each step; in addition the equations are decoupled and can therefore be solved directly without recourse to convergence checks.

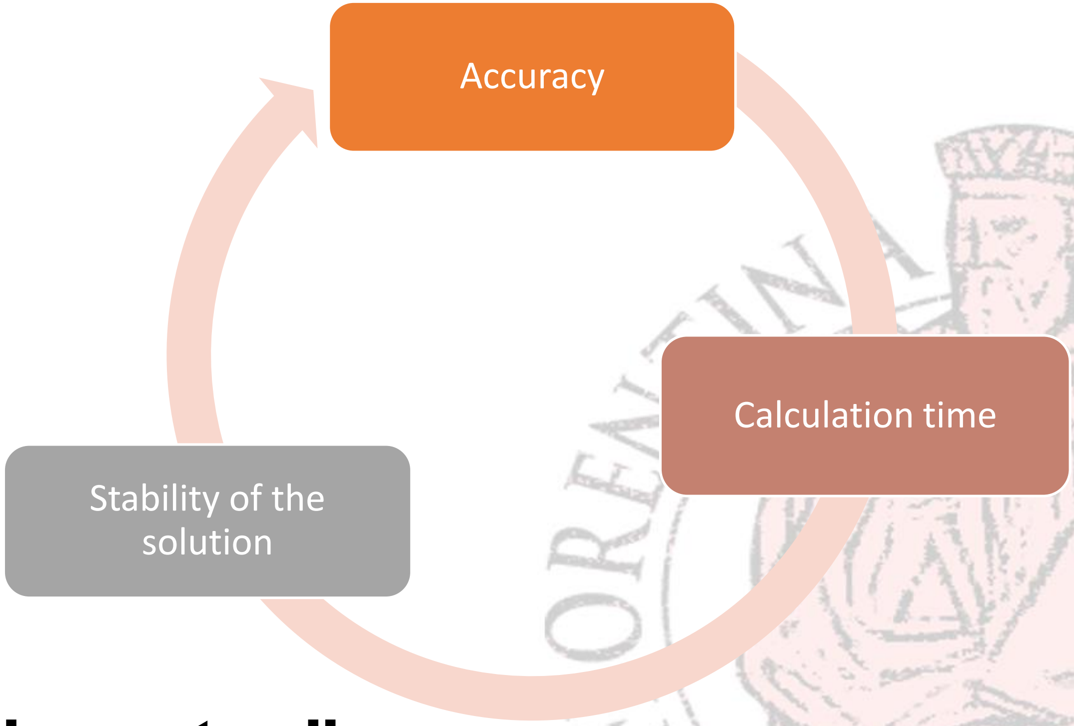
limits

The method work with very small integration intervals, which therefore quickly increase the computational cost in determining the solution, obviously seeking to achieve a sufficient accuracy.

LS DYNA Software

The main problem, in using an explicit solver like LS-DYNA in the analysis of crash phenomena, is the optimization of the three following factors:

- Accuracy;
- Calculation time;
- Stability.



Definition of the **“time step”**

LS DYNA Software

The **time step** is the integration time interval represented by the term Δt.

It depends on the <u>size of the element</u> involved in the calculation.

Sound speed in the material considered

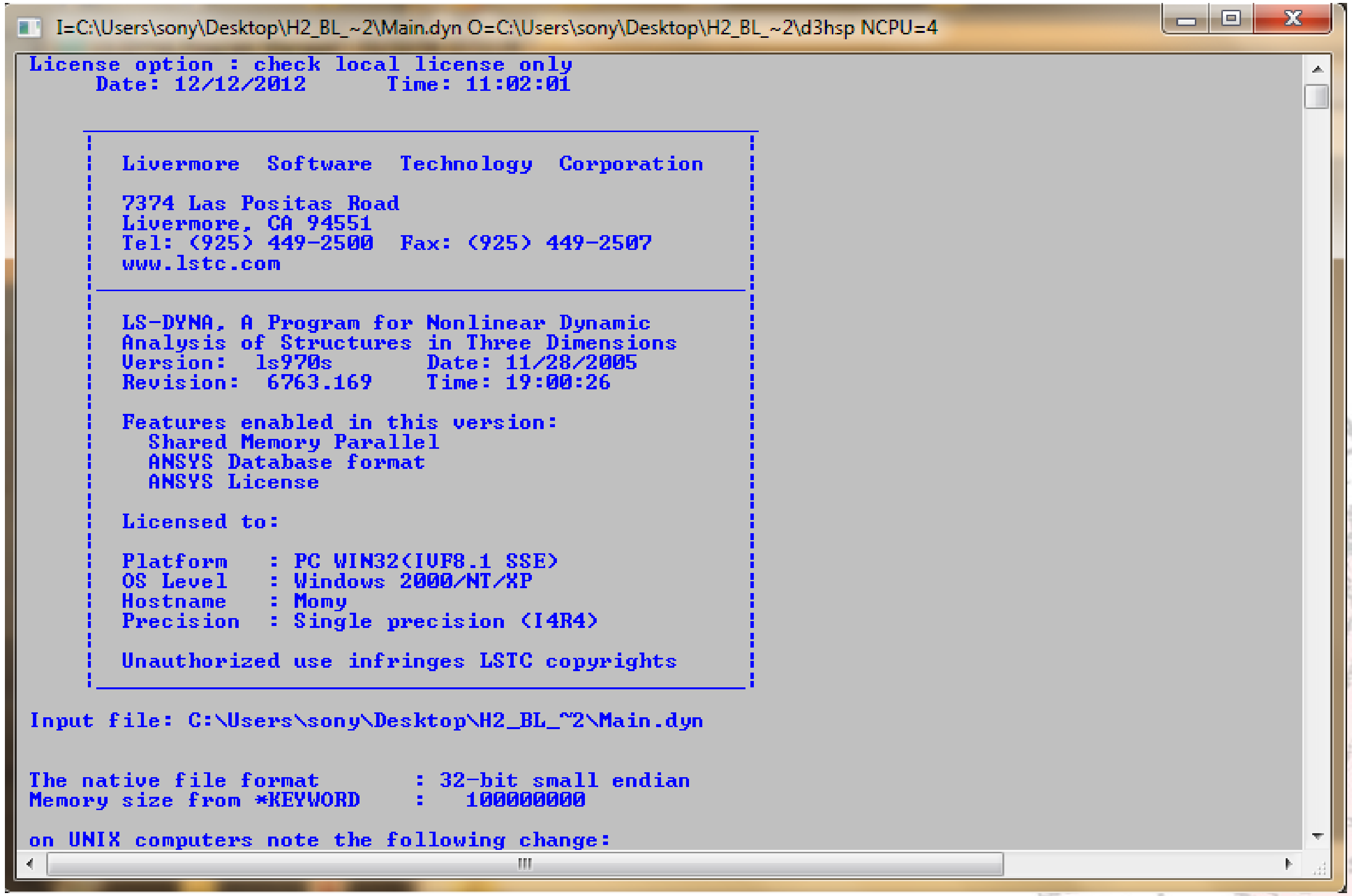
$$\Delta t = \frac{c}{l} < \Delta t, c$$

Dimesion of the element (or two node distance)

$$\Delta t_{critical} = \frac{2}{\omega_{max}} \quad \Rightarrow \quad \omega_{max} = \frac{2}{l}\, c$$

LS DYNA Software

Analysis



```
I=C:\Users\sony\Desktop\H2_BL_~2\Main.dyn O=C:\Users\sony\Desktop\H2_BL_~2\d3hsp NCPU=4

License option : check local license only
     Date: 12/12/2012       Time: 11:02:01

     Livermore  Software  Technology  Corporation

     7374 Las Positas Road
     Livermore, CA 94551
     Tel: (925) 449-2500  Fax: (925) 449-2507
     www.lstc.com

     LS-DYNA, A Program for Nonlinear Dynamic
     Analysis of Structures in Three Dimensions
     Version:  ls970s        Date: 11/28/2005
     Revision:  6763.169     Time: 19:00:26

     Features enabled in this version:
       Shared Memory Parallel
       ANSYS Database format
       ANSYS License

     Licensed to:

     Platform    : PC WIN32(IVF8.1 SSE)
     OS Level    : Windows 2000/NT/XP
     Hostname    : Momy
     Precision   : Single precision (I4R4)

     Unauthorized use infringes LSTC copyrights

Input file: C:\Users\sony\Desktop\H2_BL_~2\Main.dyn

The native file format        : 32-bit small endian
Memory size from *KEYWORD     :   100000000

on UNIX computers note the following change:
```

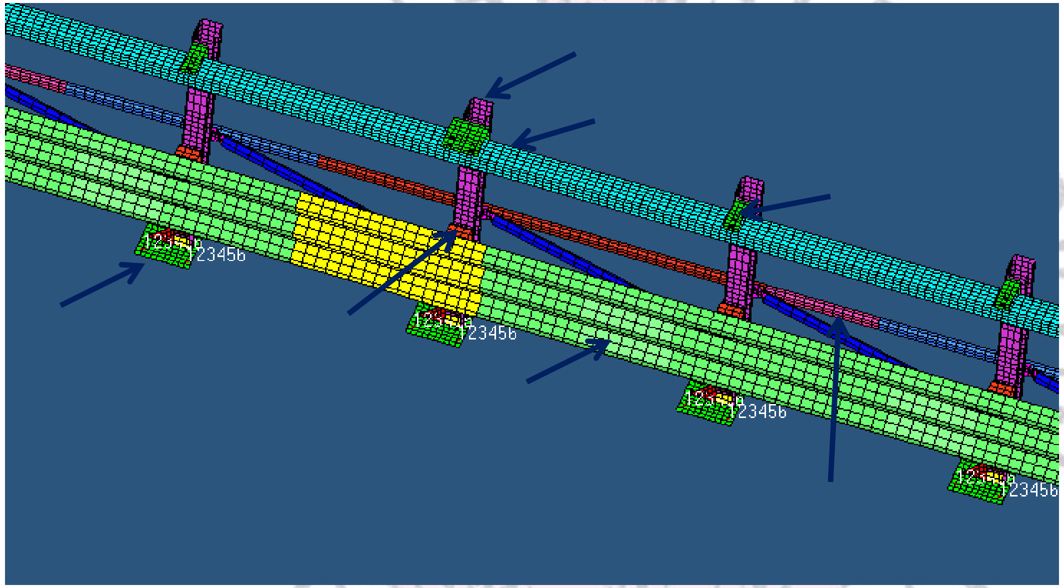
Pre-processing

Post-processing

Pre-processing

FE modeling

Definition of the geometry → FEM characterization → Assembly of the different parts



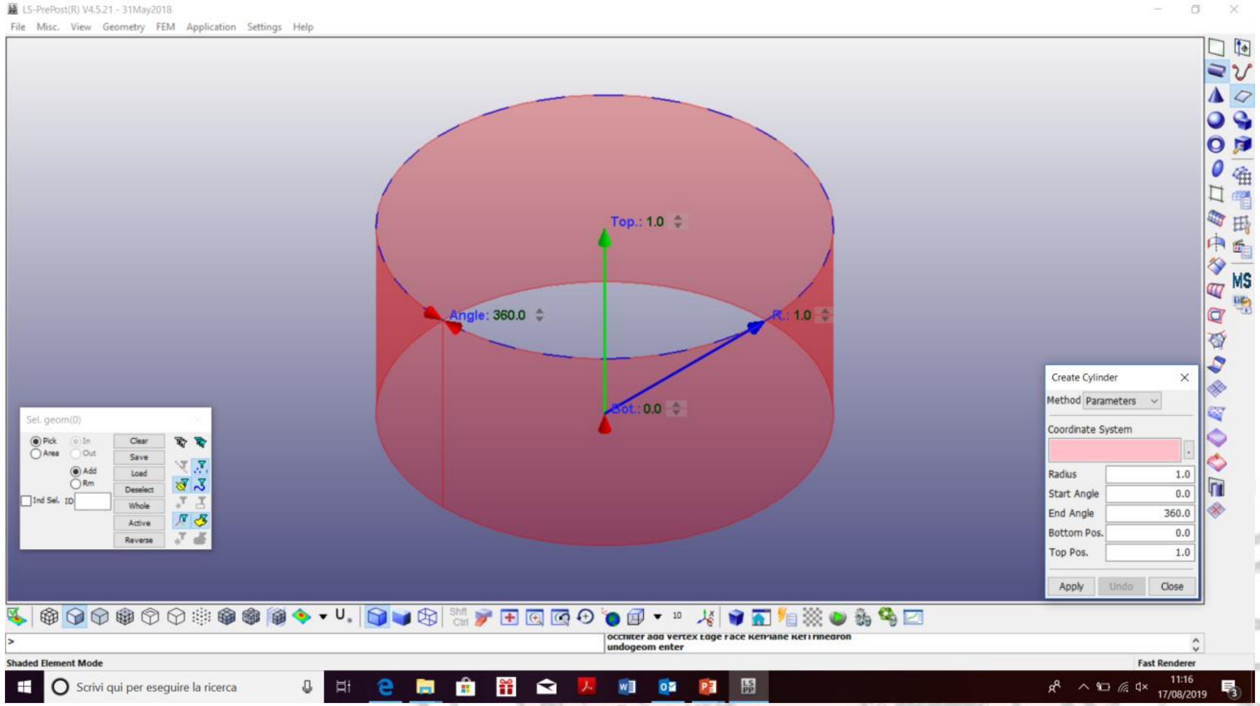
LS DYNA Software

Pre-processing

Definition of the geometry

Construction of the 3D model/models

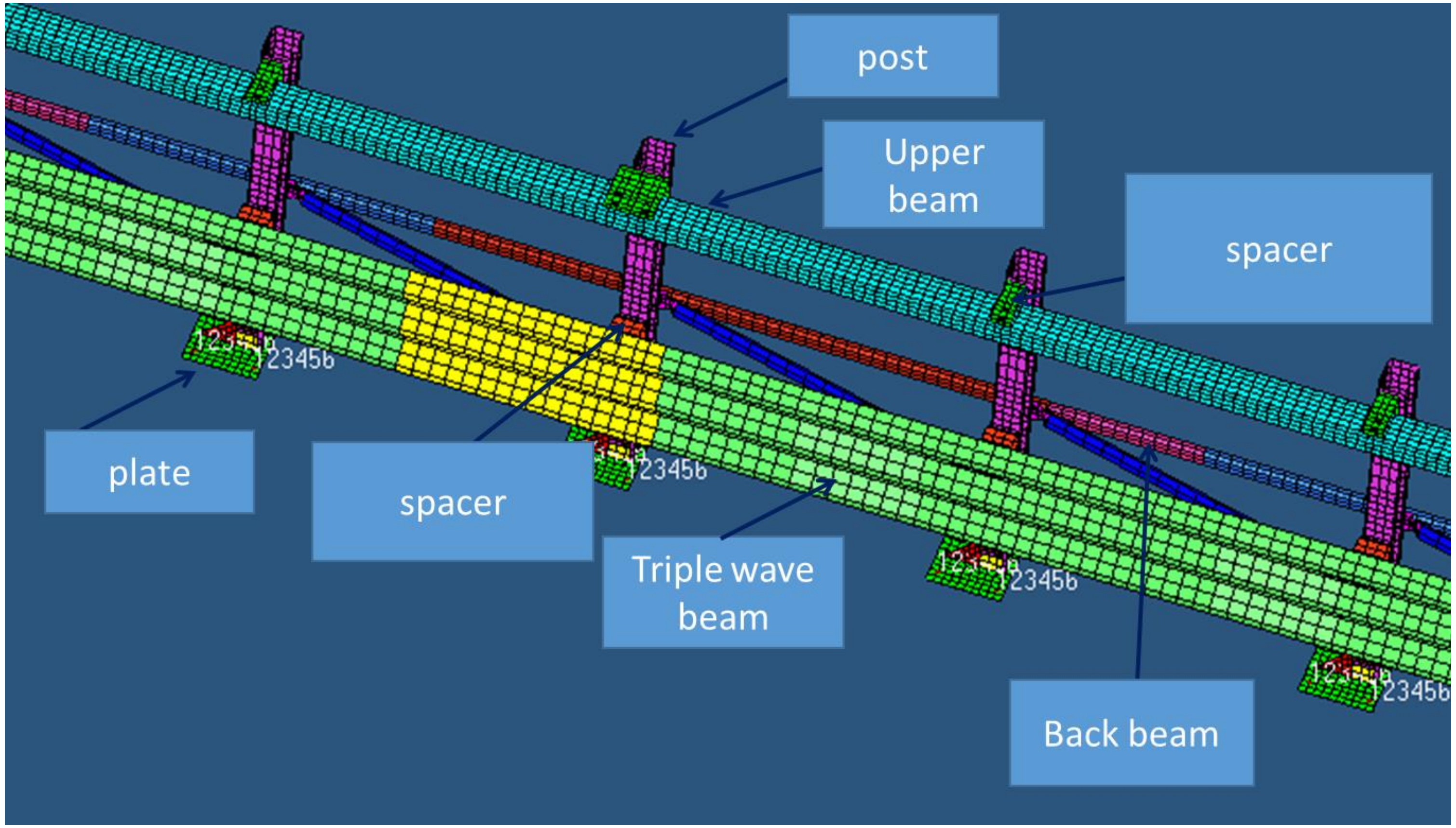
Surface modeling → Mid surface



LS DYNA Software

Pre-processing

FEM characterization



Hierarchical approach

*part

*section + *mat

database

LS DYNA Software

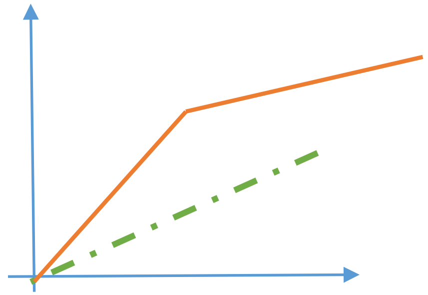
Pre-processing

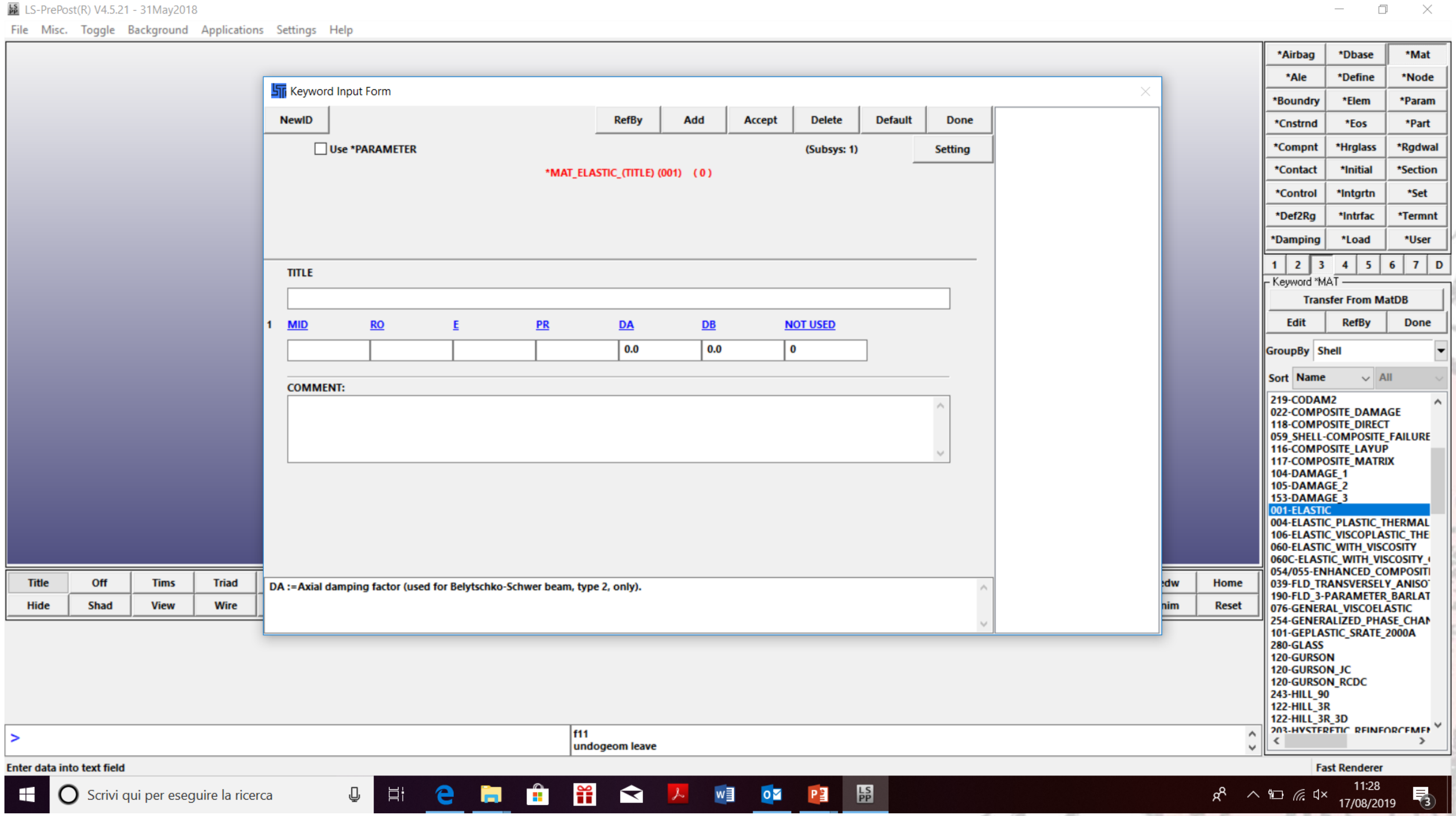
FEM characterization

*MAT_24 (MAT_PIECEWEISE_LINEAR_PLASTICITY)

*mat

001 elastic





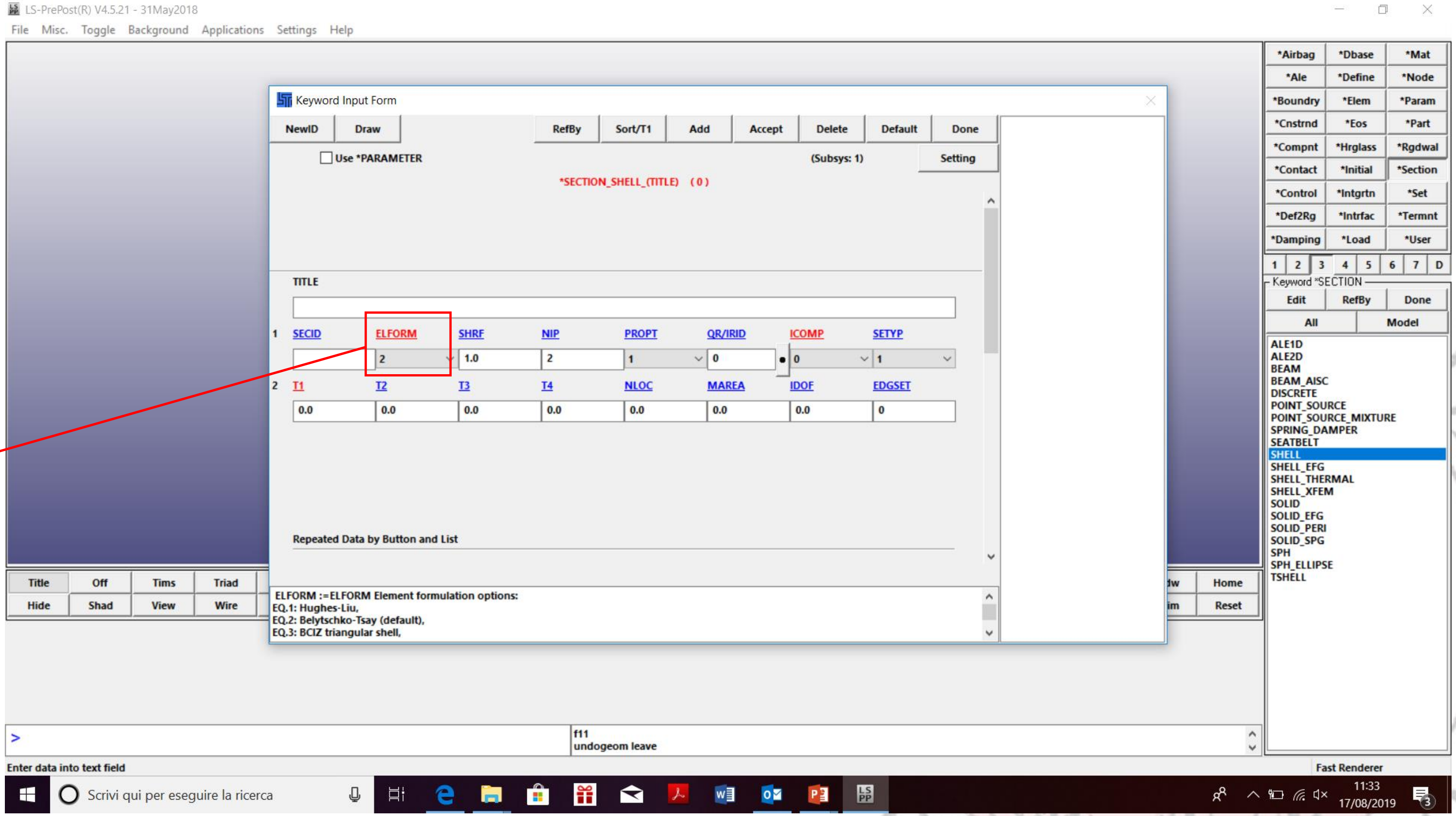
LS DYNA Software

Pre-processing

FEM characterization

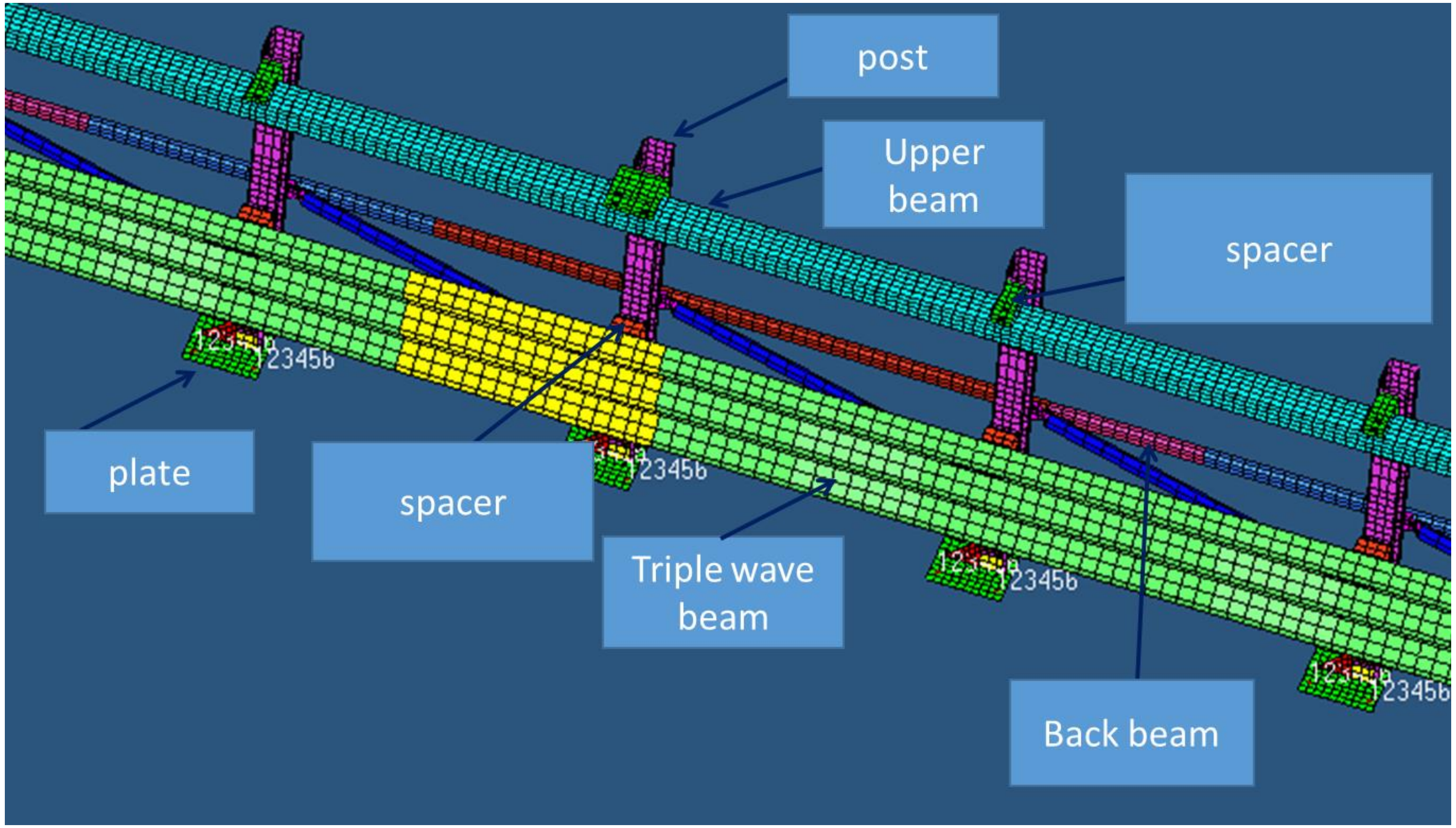
*section

Type of element and # of integration point



Pre-processing

FEM characterization



*part

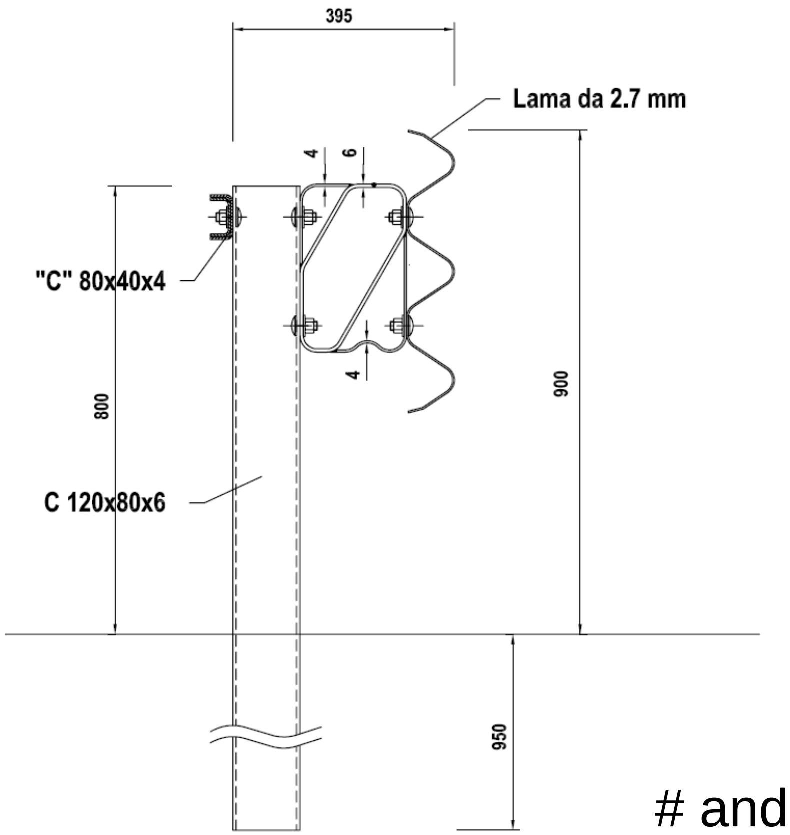
*section + *mat

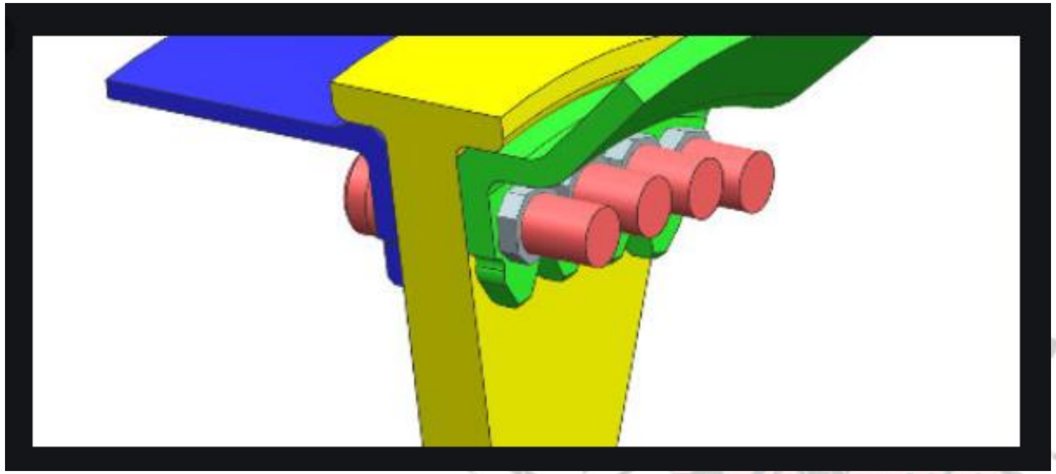
Connection between different parts of the model..and with the environment

LS DYNA Software

Pre-processing

FEM characterization



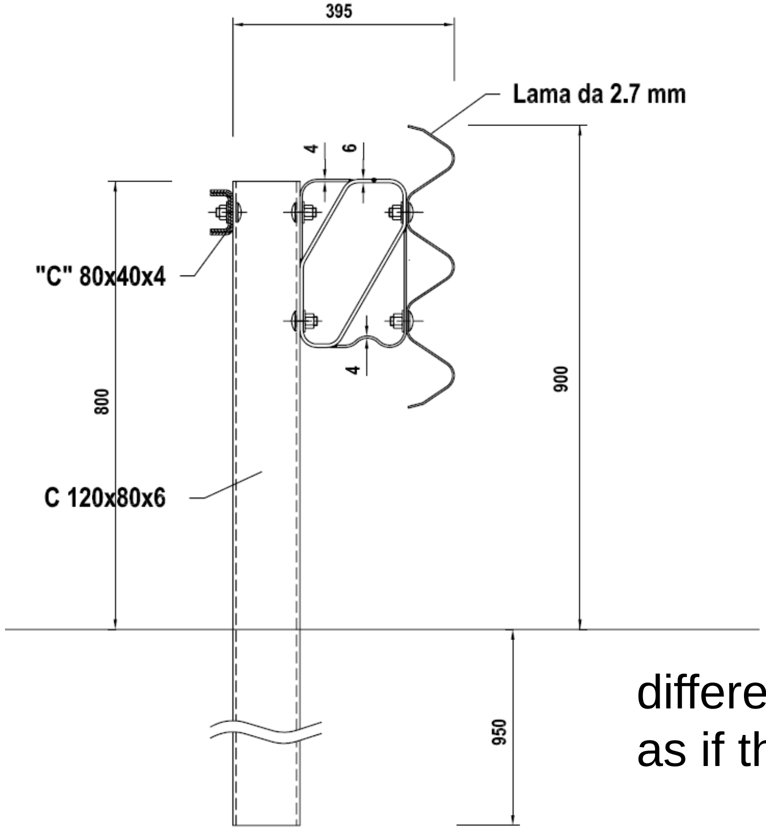


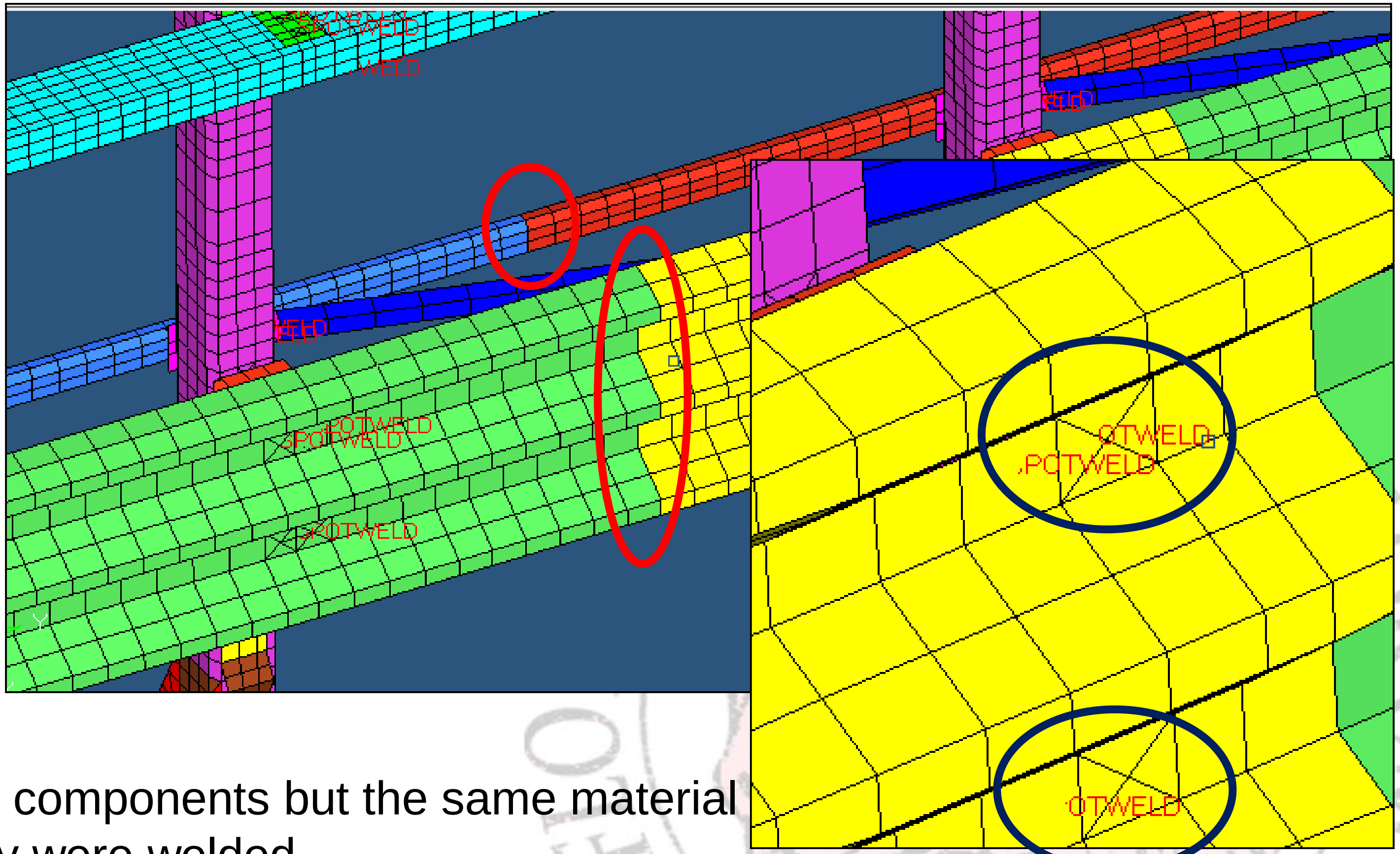
Connection between different parts of the model → C/B analysis will be conducted

and dimensions of the elements → computational cost and findings

Pre-processing

FEM characterization





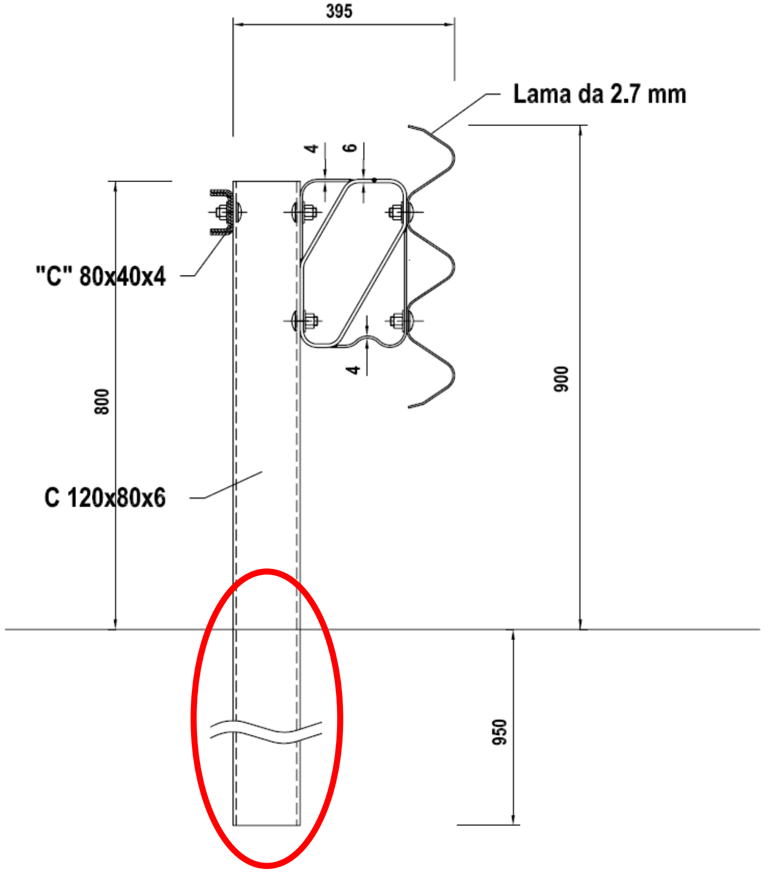
different components but the same material as if they were welded

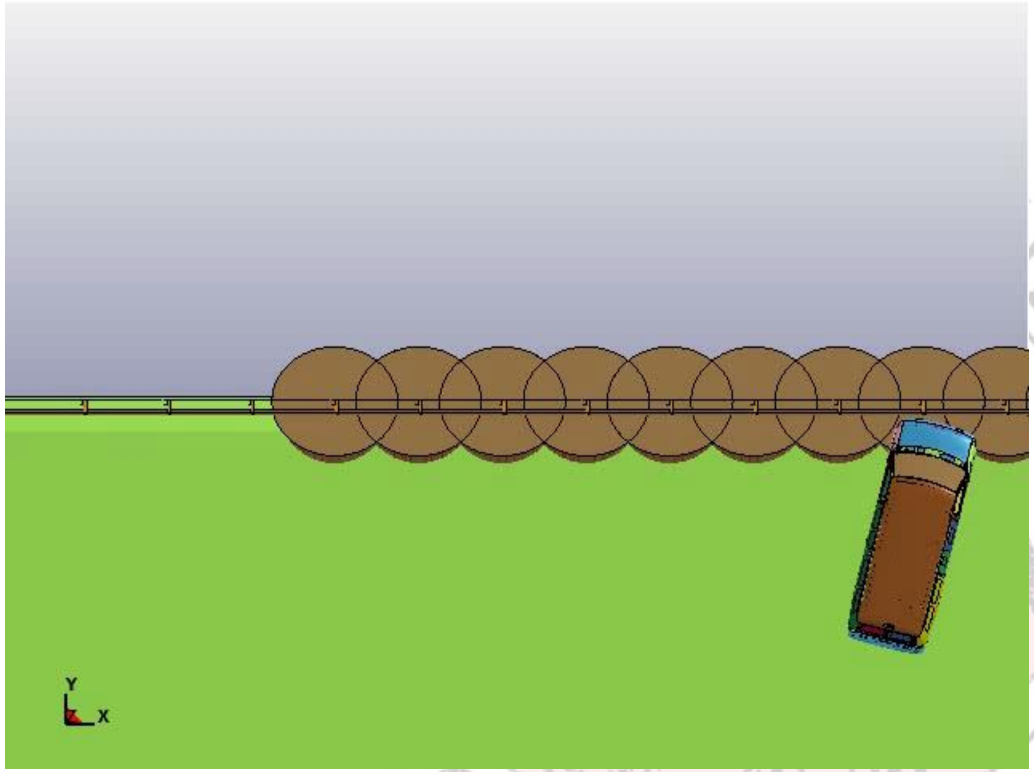
1-D elements → characterized by the same property of the bolts

Pre-processing

FEM characterization

1) Soil modelling → solid element in order to reproduce the real effect





Pre-processing

FEM characterization

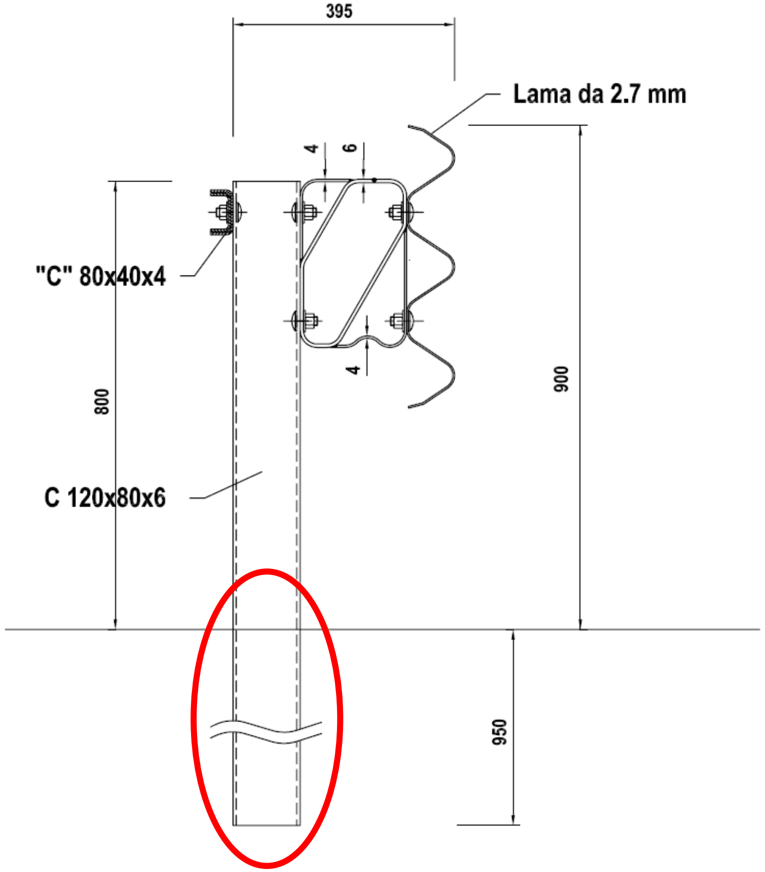
1) Soil modelling → solid element in order to reproduce the real effect

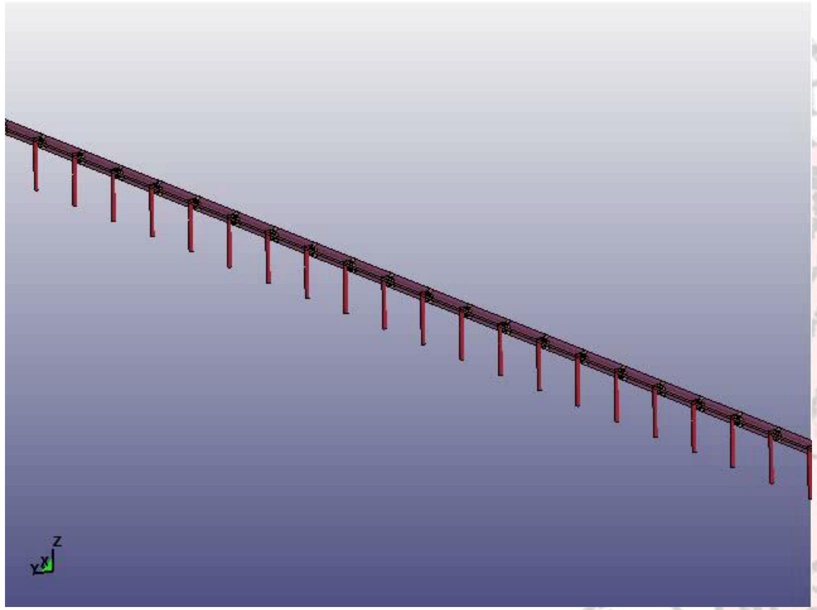
Pre-processing

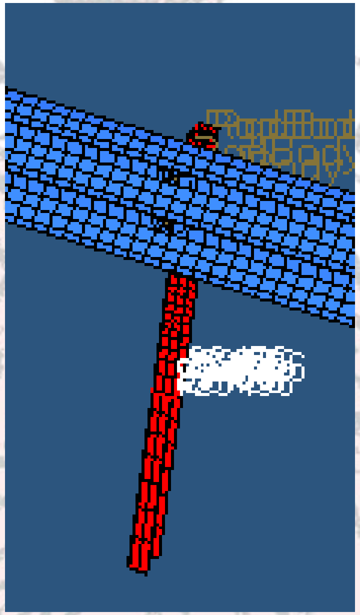
FEM characterization

2) Definition of boundary condition

6 DoF → x,y,z directions and 3 rotations

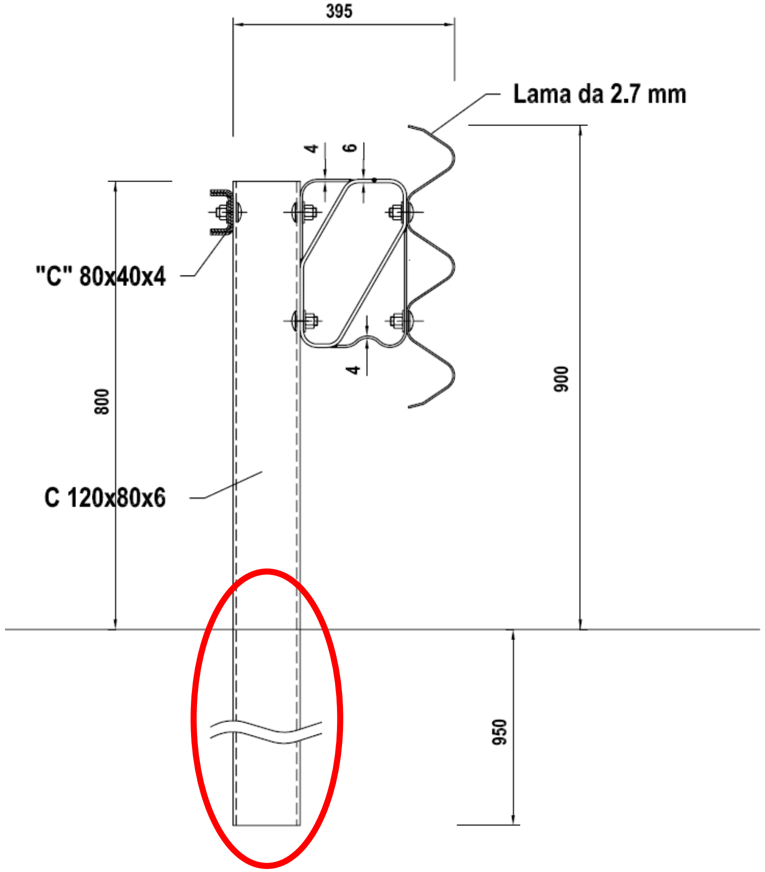






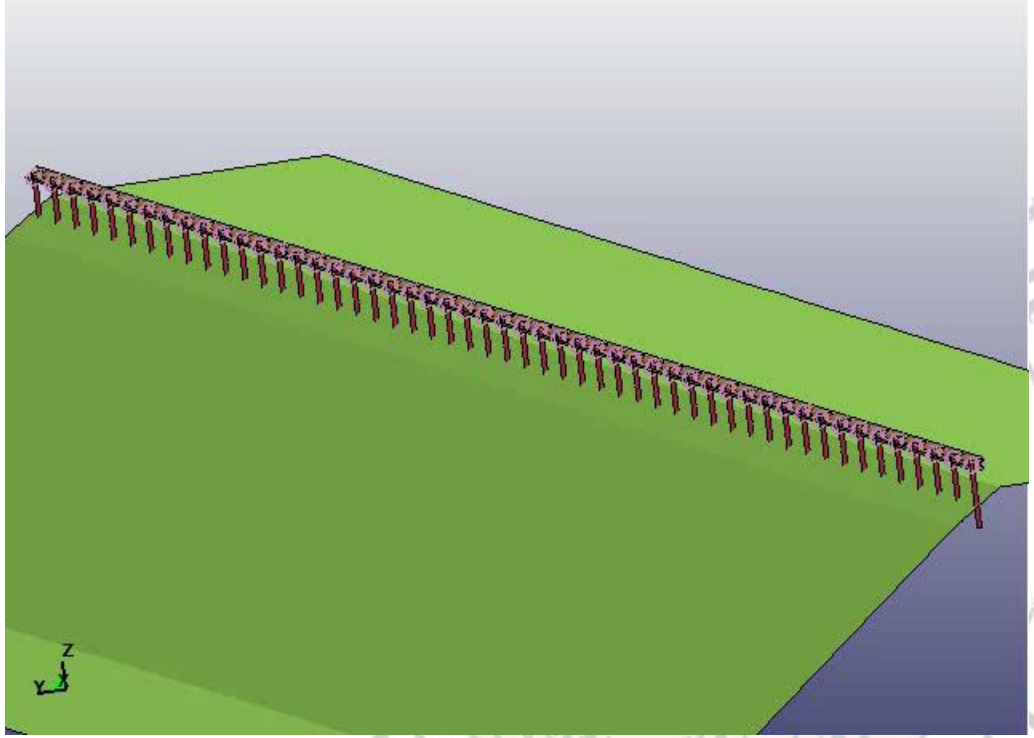
Pre-processing

FEM characterization



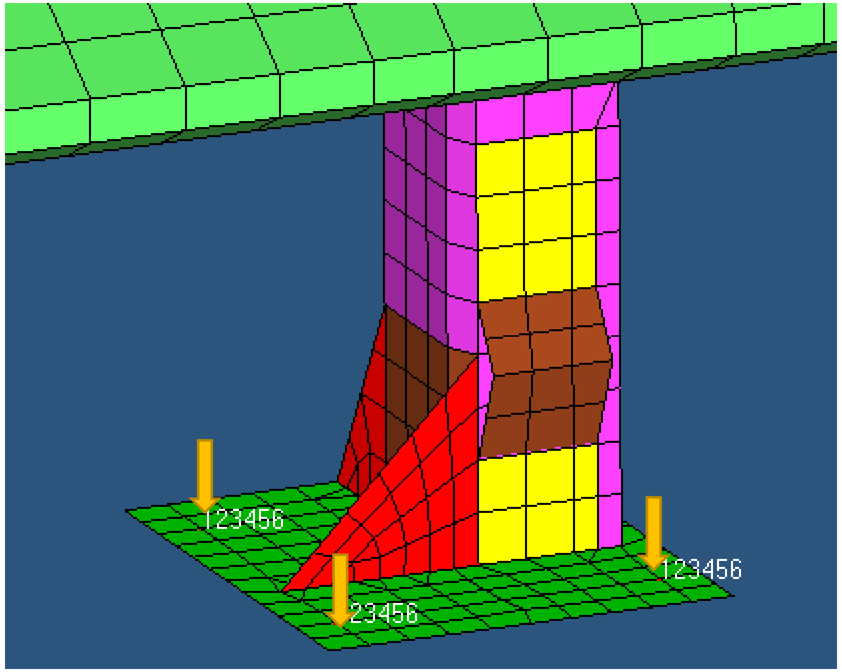
2) Definition of boundary condition

6 DoF → x,y,z directions and 3 rotations

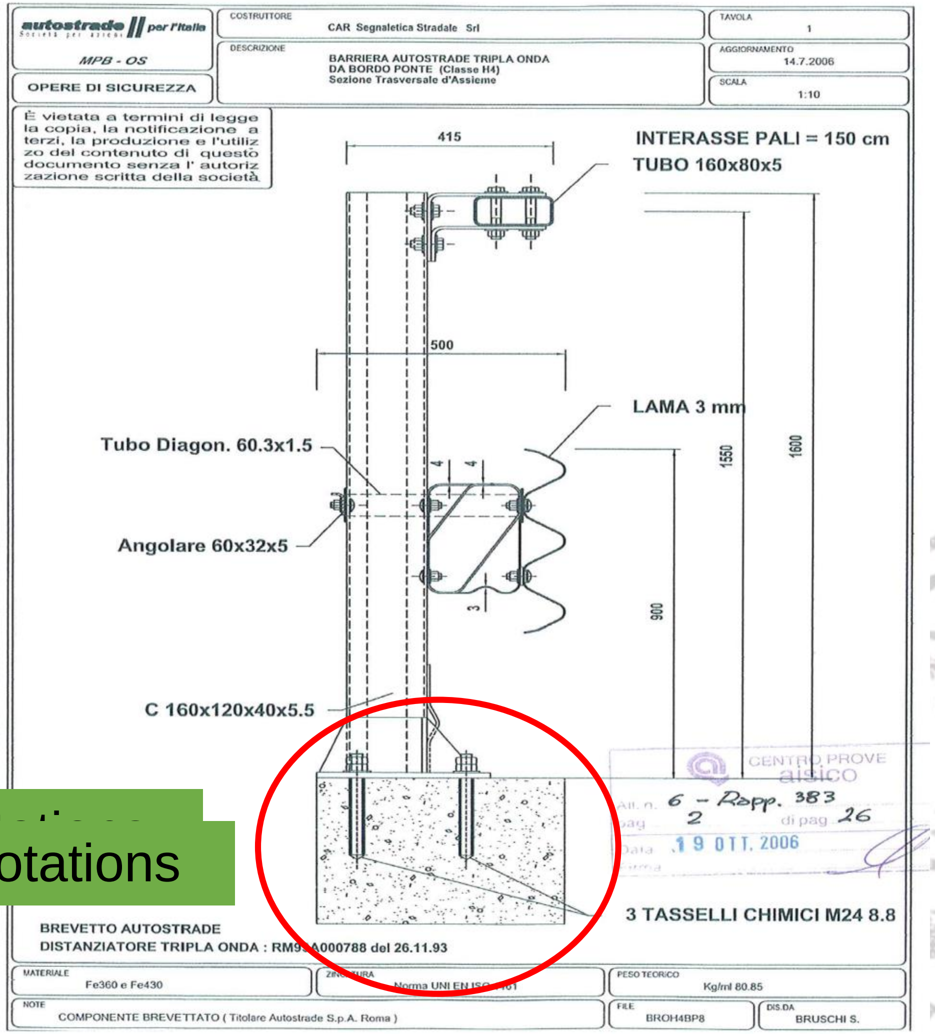


Pre-processing

FEM characterization

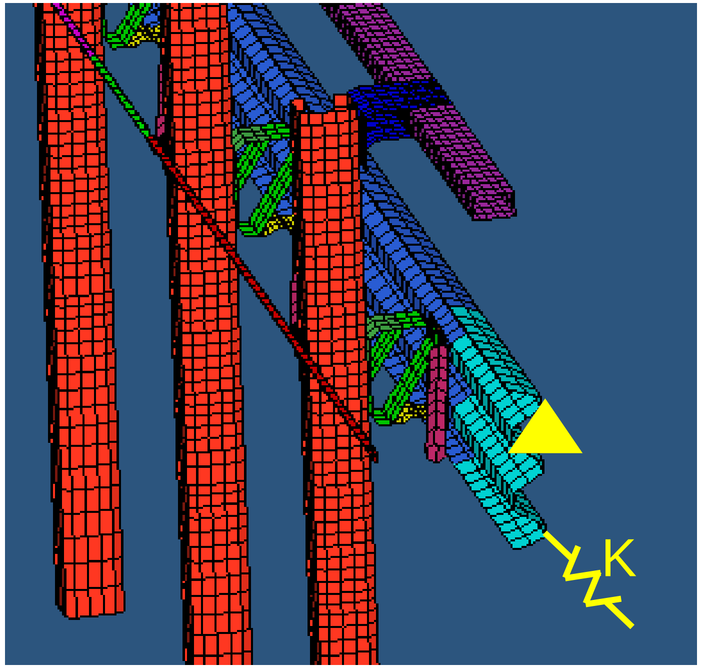


No translations/no rotations



Pre-processing

FEM characterization



The selection of the type of BCs depends:
1) from the behaviour of the barrier during the crash test;
2) from the behaviour of the barrier during the accident.

the total length of the device also affects the selection of the constraint

…and what are the BCs at the end of the barrier? how is the terminal modeled?

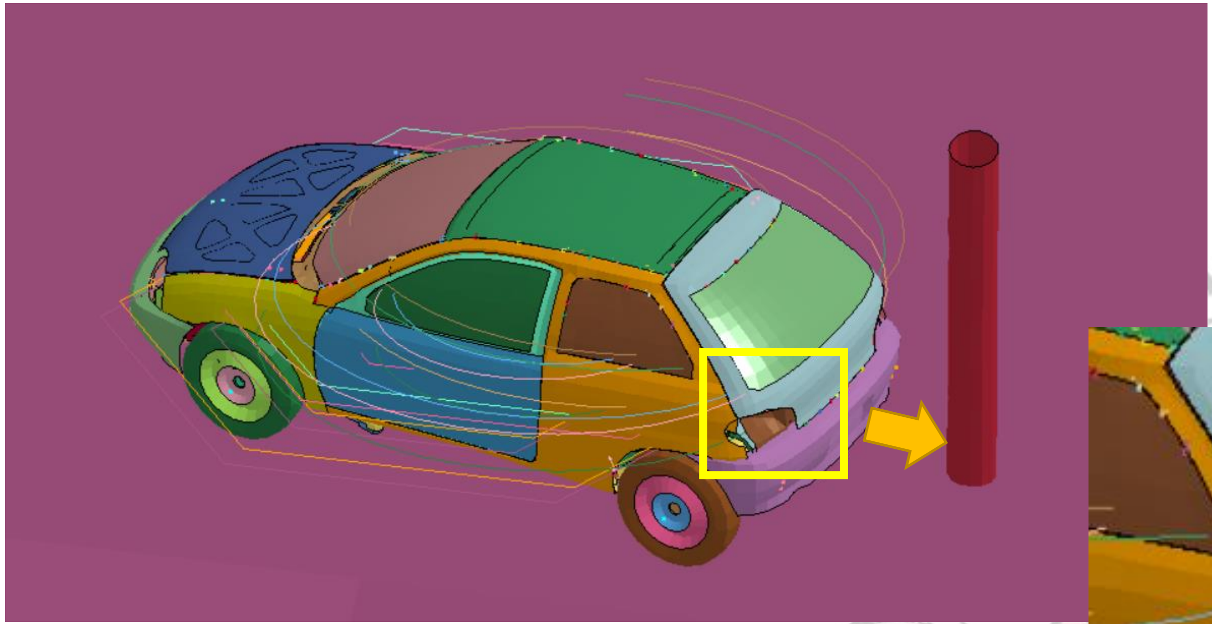
Pre-processing

Contact → Interaction between two (or more) different object

Implicit VS Explicit

ANSYS LS DYNA

high simplicity of contact setup



contact management is necessary both to represent the crash phenomena and to represent the interaction between two parts of the same "object"

Pre-processing

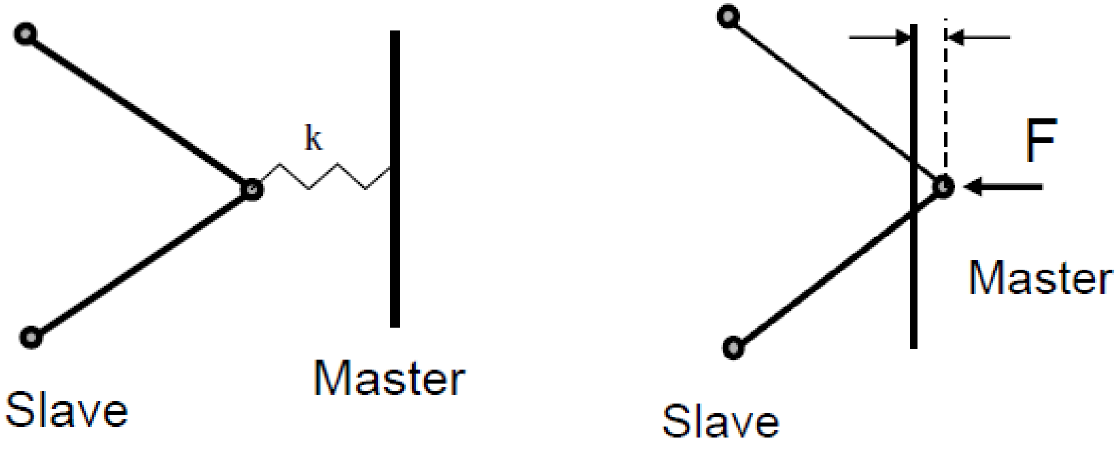
Contact → Interaction between two (or more) different object

- Kinematic costraint Method;
- Penalty method;
- Distribuited Parameter Method.

Pre-processing

Contact → Penalty methods

Main contact used for the reproduction of the crash phenomena



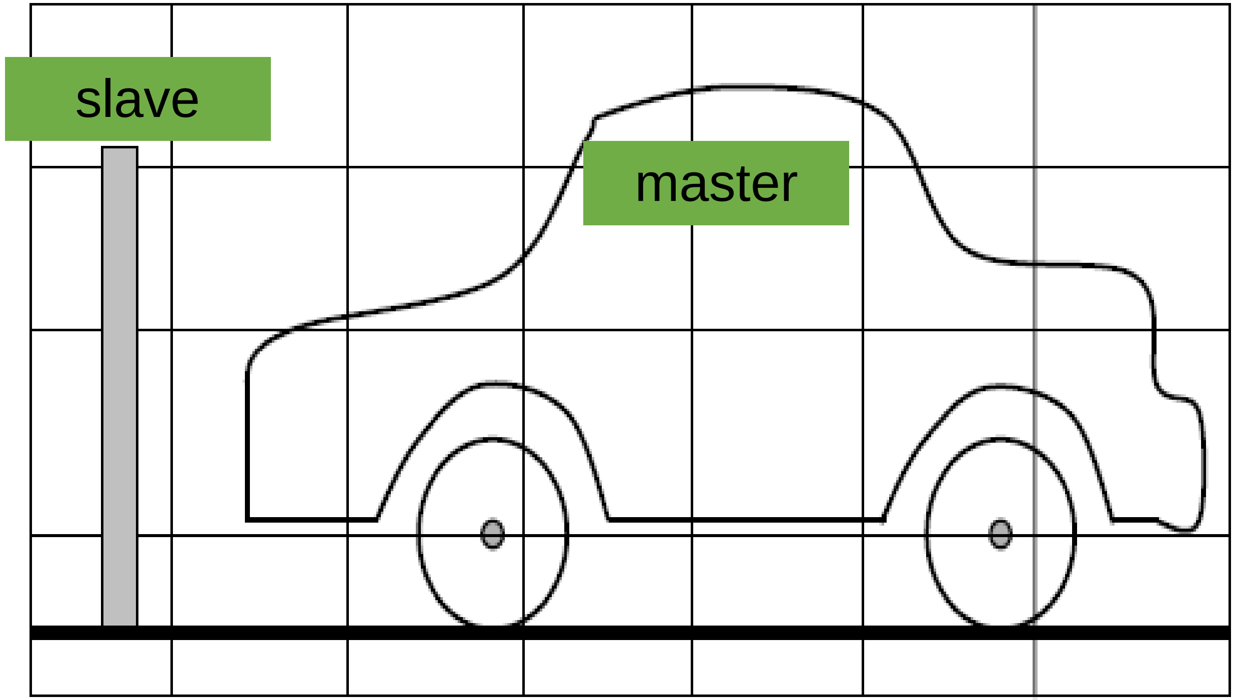
slave

master

1. Contact Search;
2. Contact Orientation;
3. Offset of the affected surfaces;
4. Contact Stiffness.

Pre-processing

Contact search



Practically , after the user has chosen the elements involved in the contact, the solver builds a grid and verify the distance between each element of the grid separately, without considering those that are far apart.

Advantages: reduction of computational cost

Pre-processing

Contact

- Kinematic costraint Method;
- Penalty method;
- Distribuited Parameter Method.

For certain types of contact, such as pure scrolling, the penalty factor method can lead to very long computational times

It imposes constraints to global equations

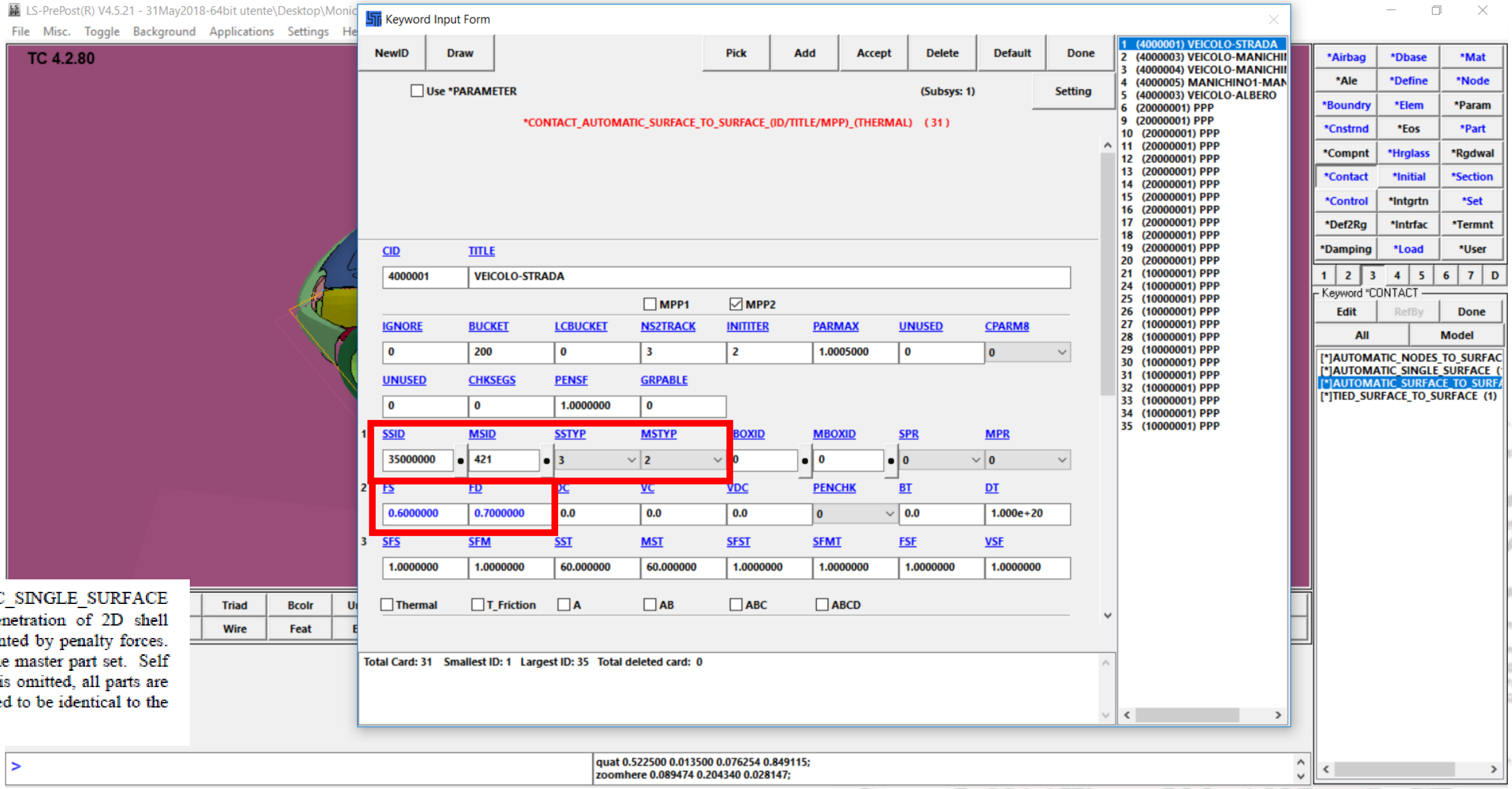
- Nodal Rigid Body Costraint (Vincolo rigido)
- Spot-Welds (Punto di saldatura)
- Joints (Giunti)

They differ mainly in the type of constraint offered (number of blocked degrees of freedom) and in the ability to provide or not to provide a break criterion… i.e. spotweld

$$\left(\frac{|f_n|}{S_n}\right)^2 + \left(\frac{|f_s|}{S_s}\right)^2 \geq 1$$

Pre-processing

Contact



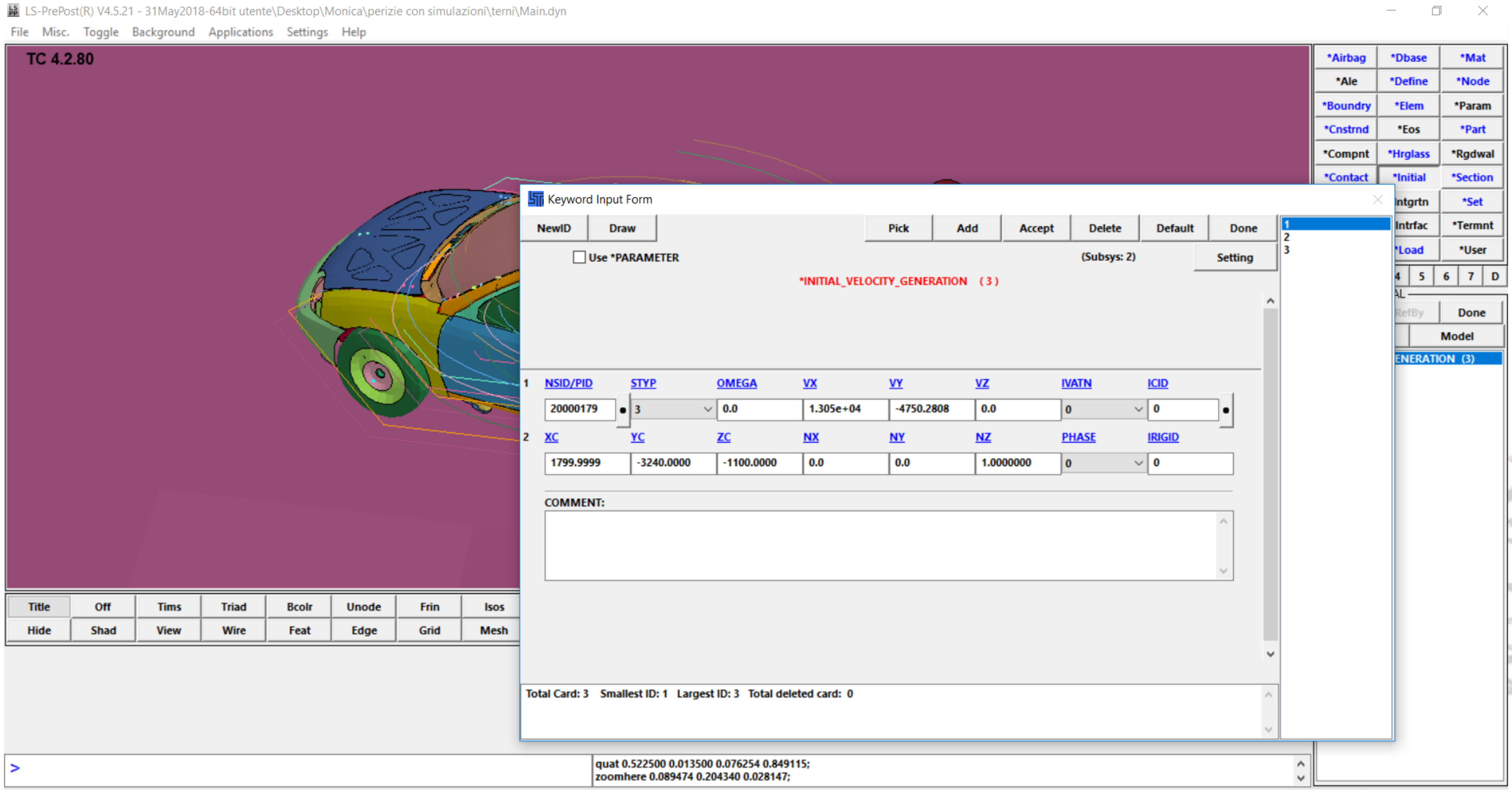
1. For AUTOMATIC_SURFACE_TO_SURFACE, AUTOMATIC_SINGLE_SURFACE contact and AUTOMATIC_NODE_TO_SURFACE contact, penetration of 2D shell elements and external faces of 2D continuum elements is prevented by penalty forces. Parts in the slave part set are checked for contact with parts in the master part set. Self contact is checked for any part in both sets. If the slave part set is omitted, all parts are checked for contact. If the master part set is omitted, it is assumed to be identical to the slave part set.

LS DYNA Software

Pre-processing

**Initial …velocity*

**Define curve…*



LS DYNA Software

…and practically….