

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

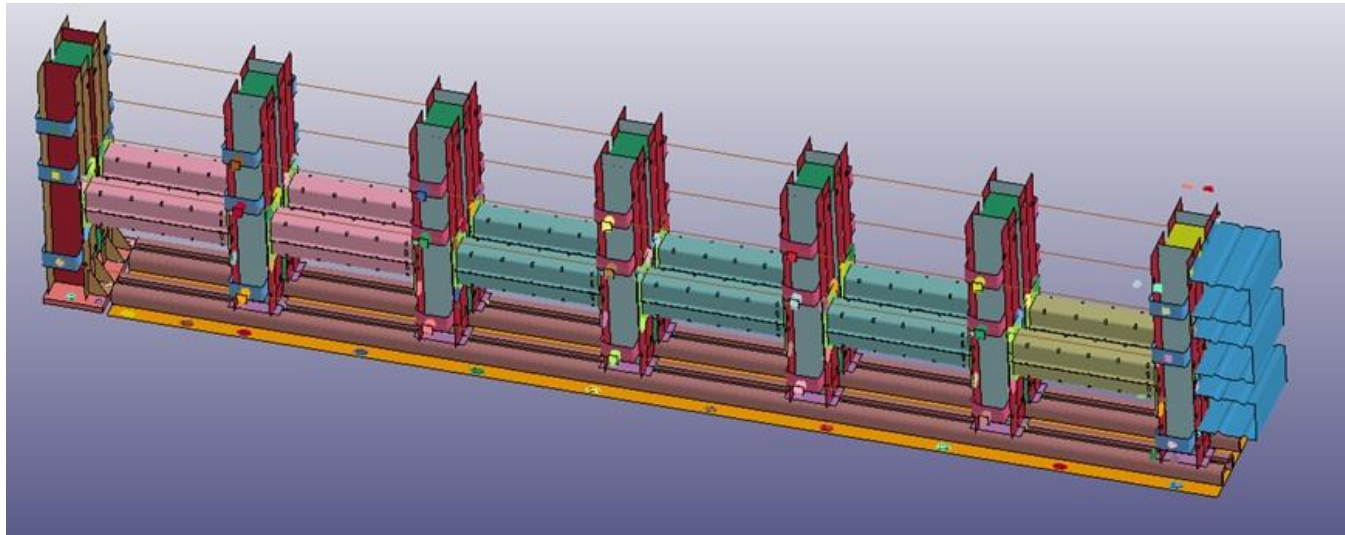
Topic Introduction

Phd. Eng. Monica Meocci

September, 09 - 2019

The FEM Methods

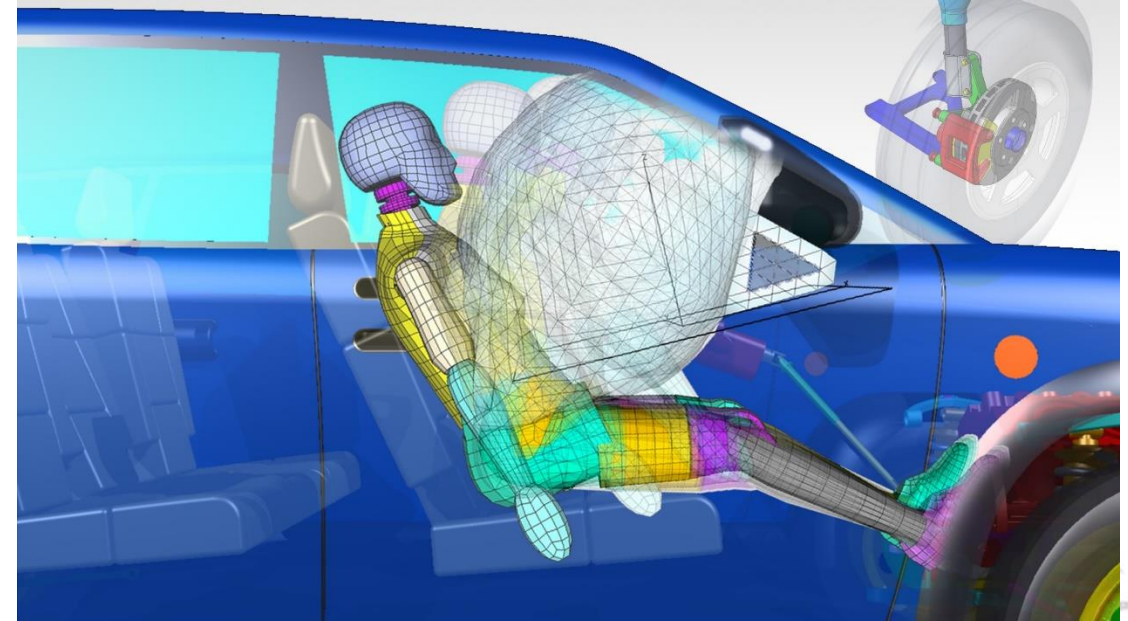
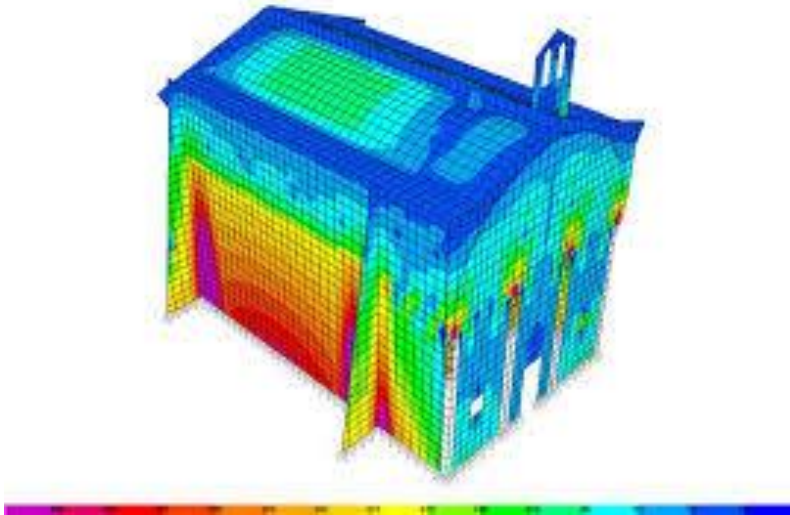
The **finite element method (FEM)** is a numerical method for solving problems of engineering and mathematical physics.



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.

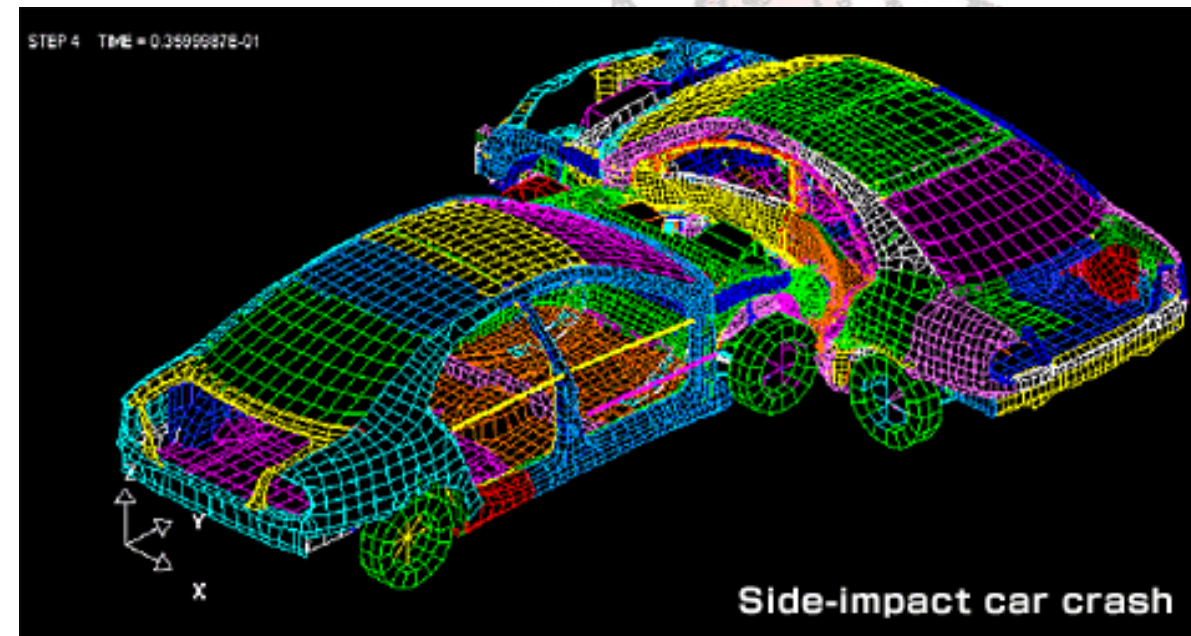


To solve the problem, it subdivides a large system into smaller, simpler parts that are called finite elements. The simple equations that model these finite elements are then assembled into a larger system of equations that models the entire problem.

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 in highly nonlinear, transient dynamic finite element analysis using explicit time integration.



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).

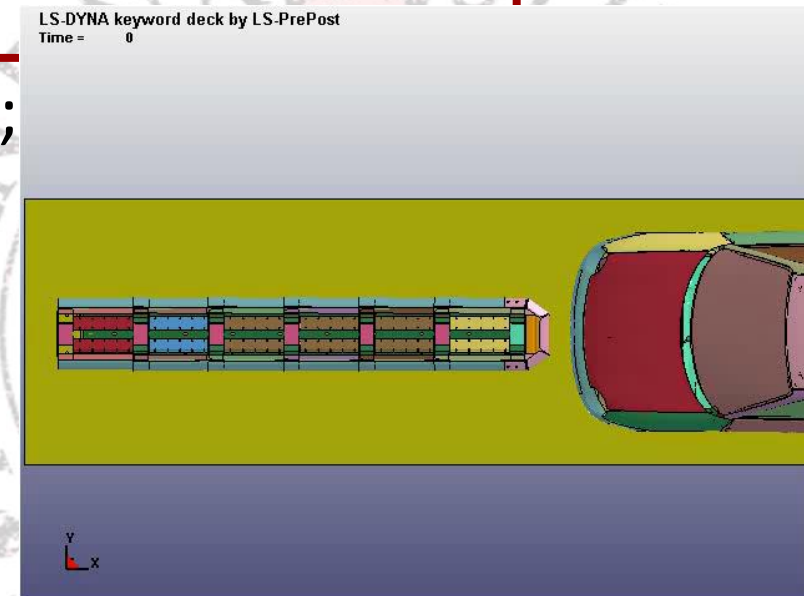


Transient dynamic

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

Typical uses include:

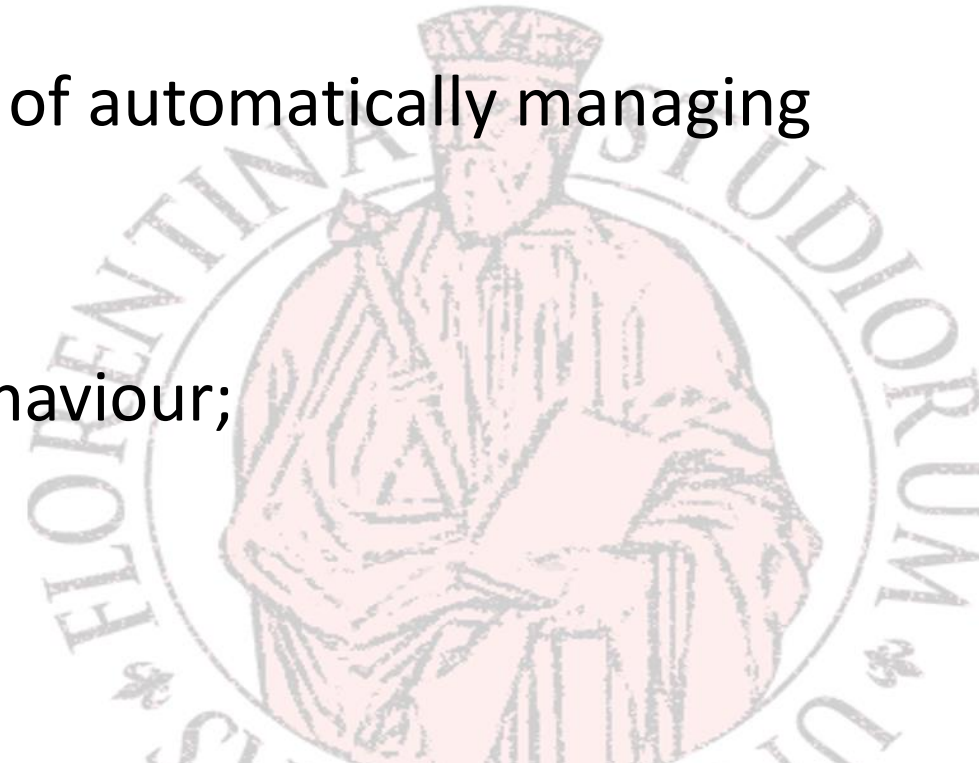
- Automotive crash (deformation of chassis, airbag inflation, seatbelt tensioning);
- Explosions (underwater Naval mine, shaped charges);
- Manufacturing (sheet metal stamping).



Need and characteristics:

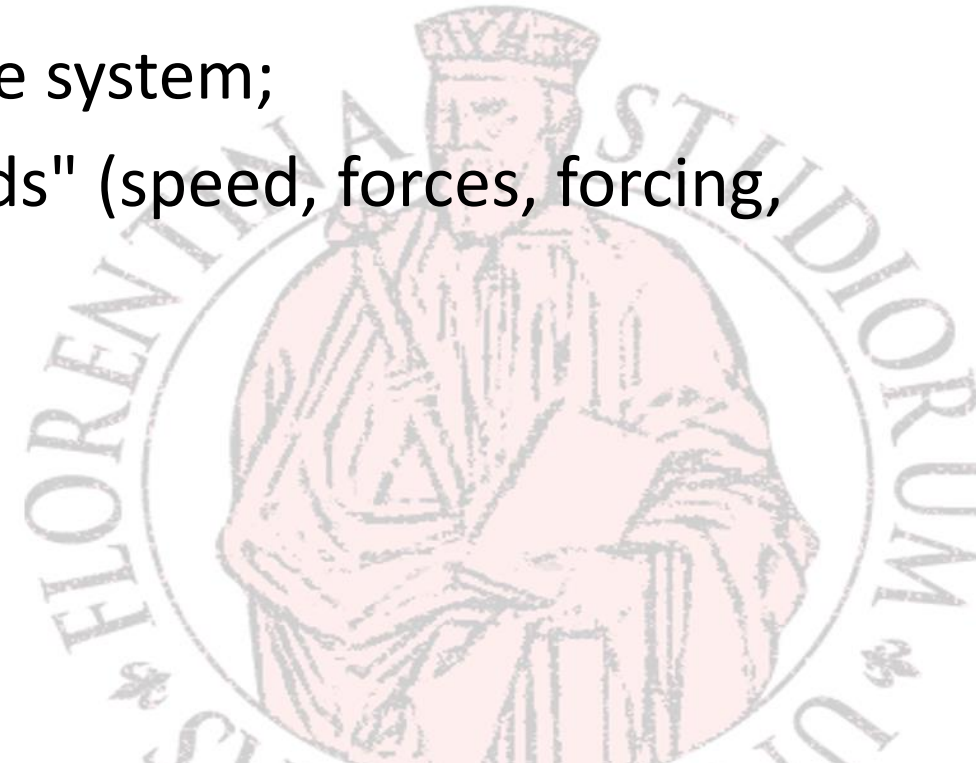
It is appropriate to investigate and solve problems characterized by:

- 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

$$\varepsilon_x = \frac{\partial u}{\partial x}$$

$$\varepsilon_y = \frac{\partial v}{\partial y}$$

$$\varepsilon_z = \frac{\partial w}{\partial z}$$

$$\gamma_{xy} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y}$$

$$\gamma_{yz} = \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}$$

$$\gamma_{zx} = \frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}$$

→ from which the internal congruence equations are derived

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

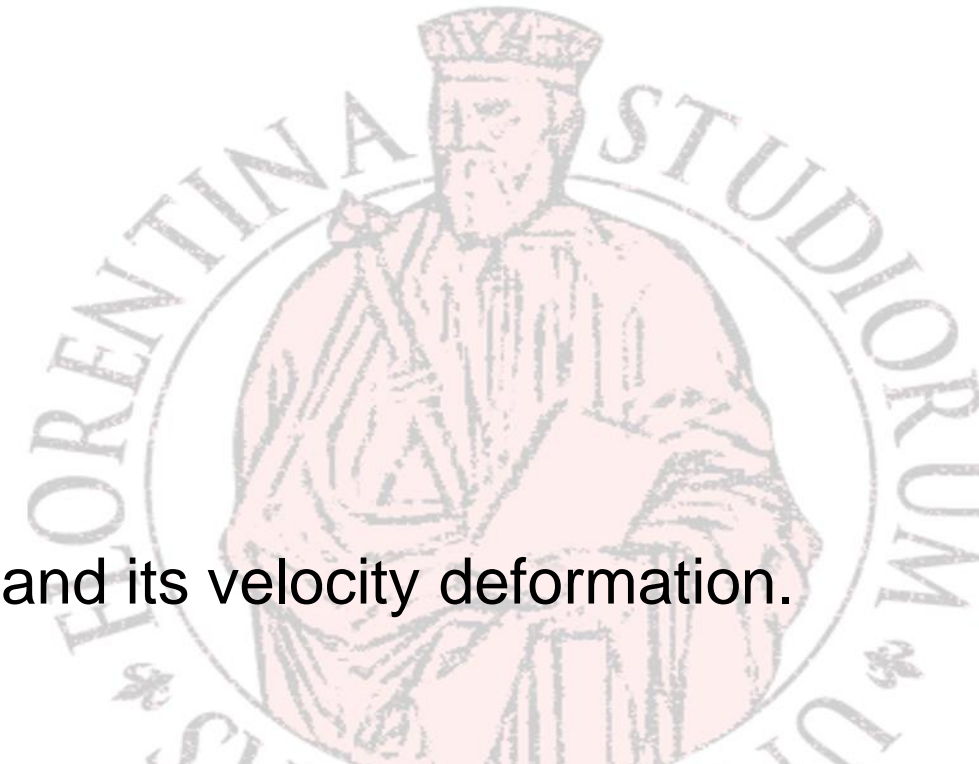
No penetration!

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

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

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.

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

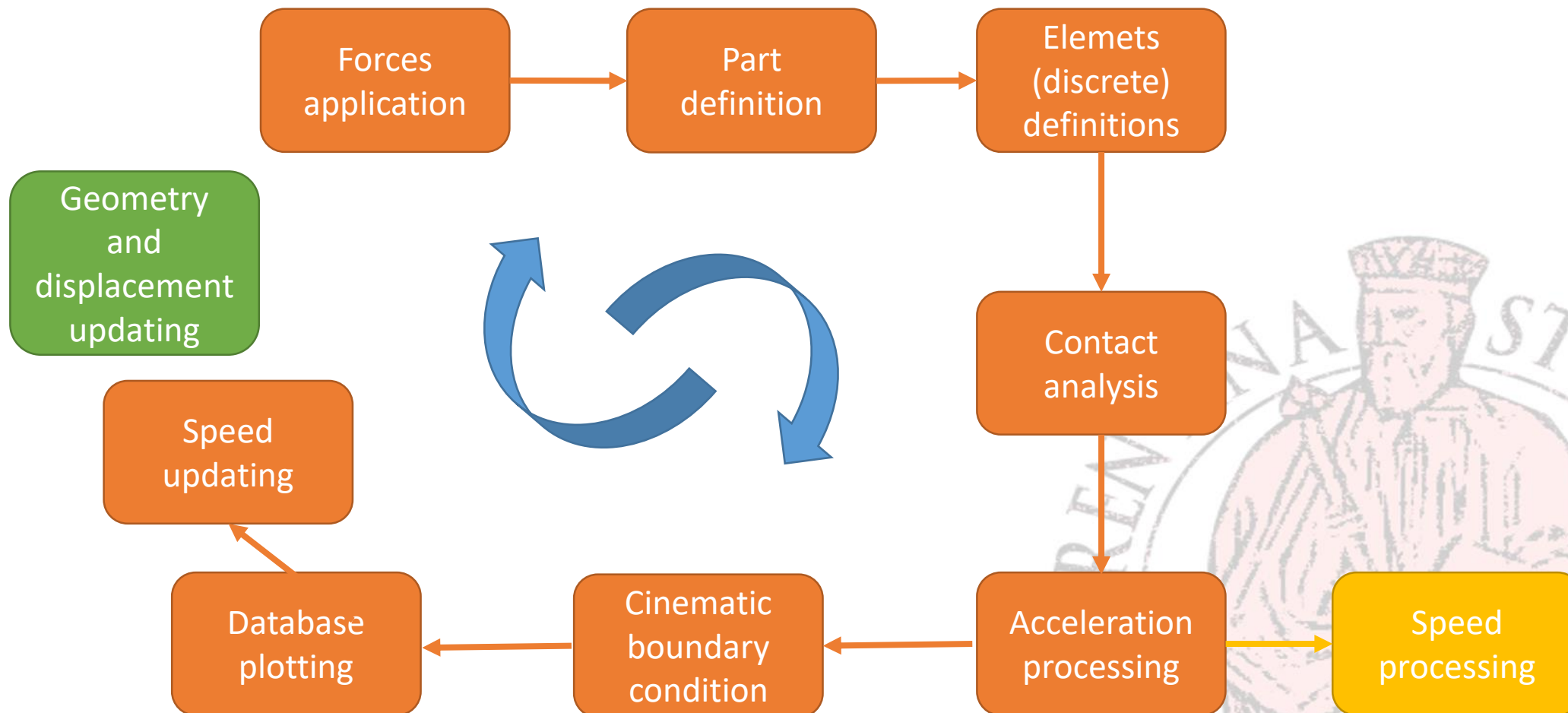
The solution to a generic time does not depend on itself,
but only on the solution at the previous instant.

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



LS DYNA Software

START →



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.

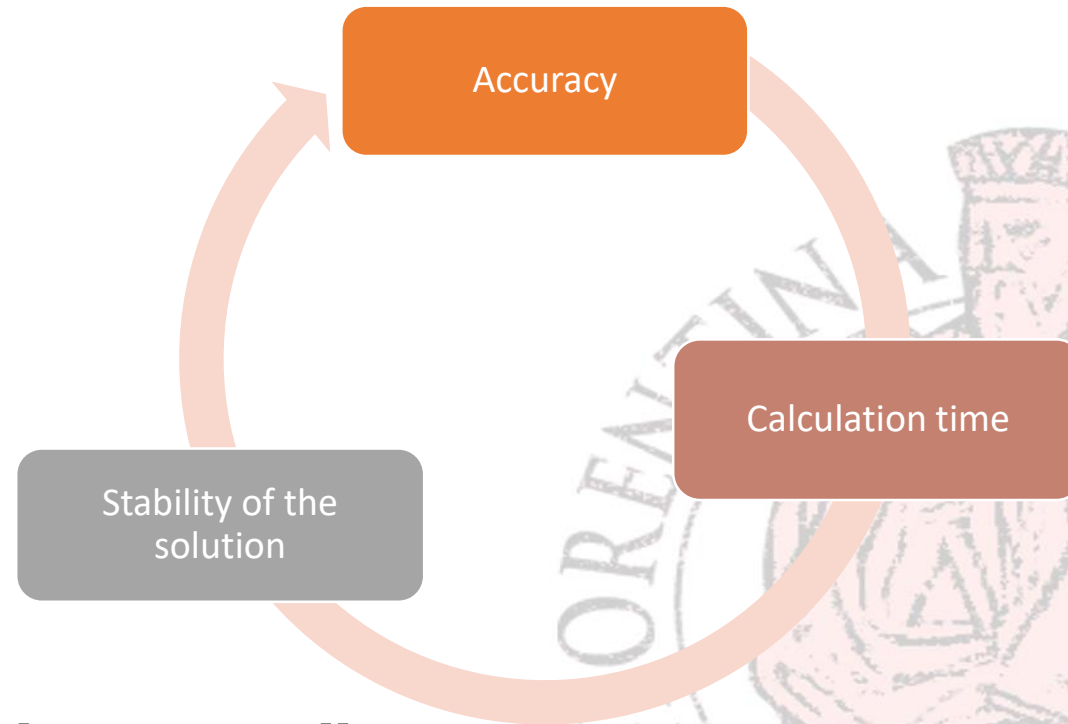
limits

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 size of the element 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)

$$\Delta t_{critical} = \frac{2}{\omega_{max}} \longrightarrow \omega_{max} = \frac{2}{l} c$$

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<IUF8.1 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

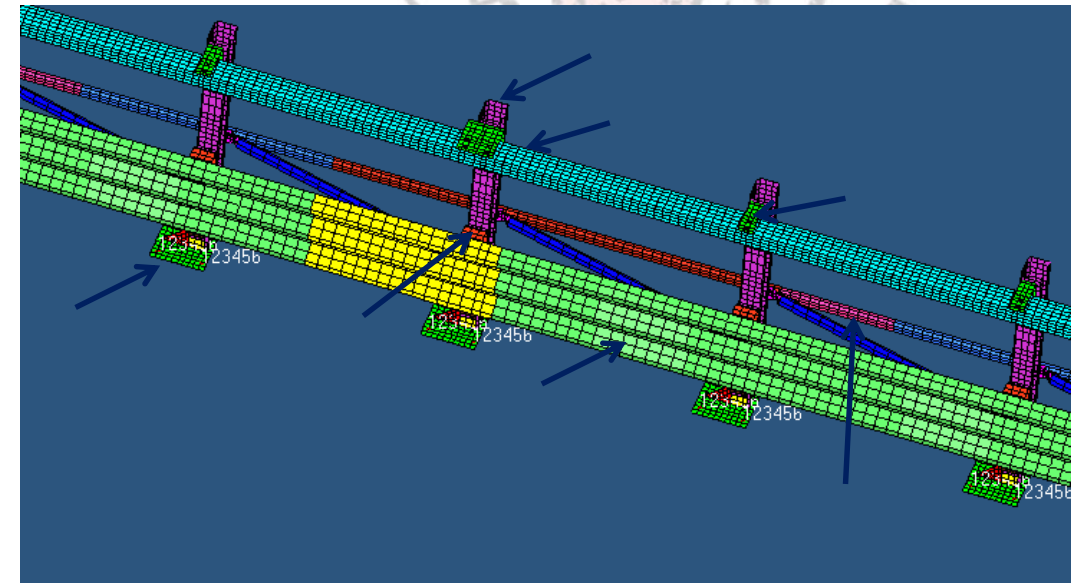
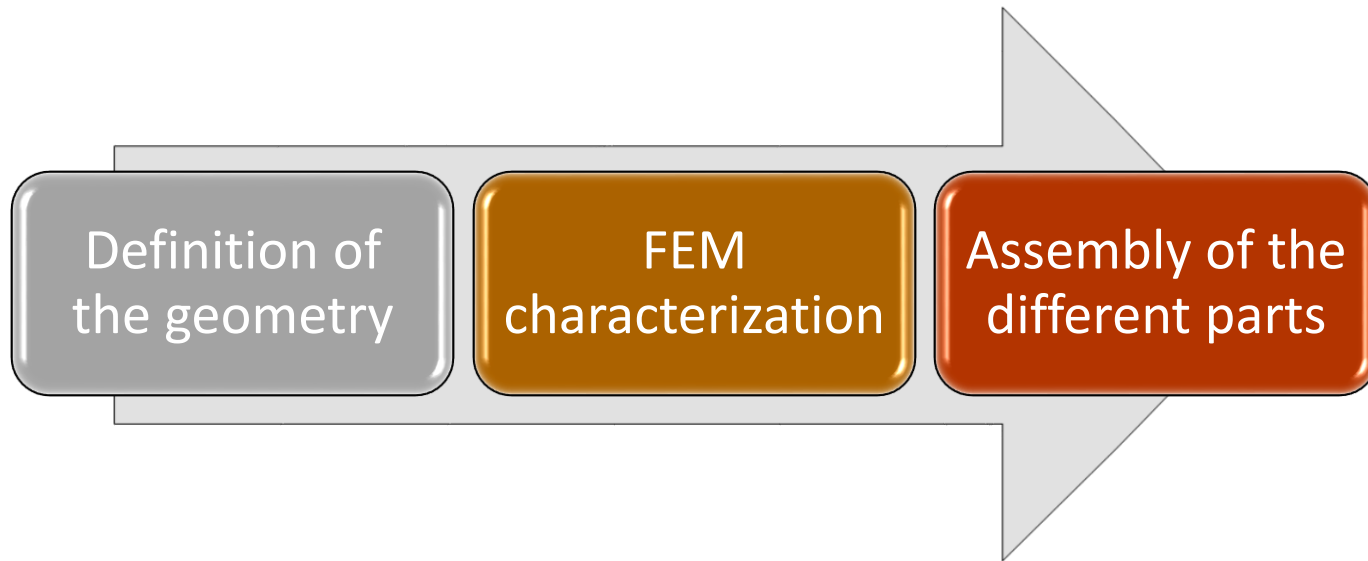
on UNIX computers note the following change:
```

Pre-processing

Post-processing

Pre-processing

FE modeling



Pre-processing

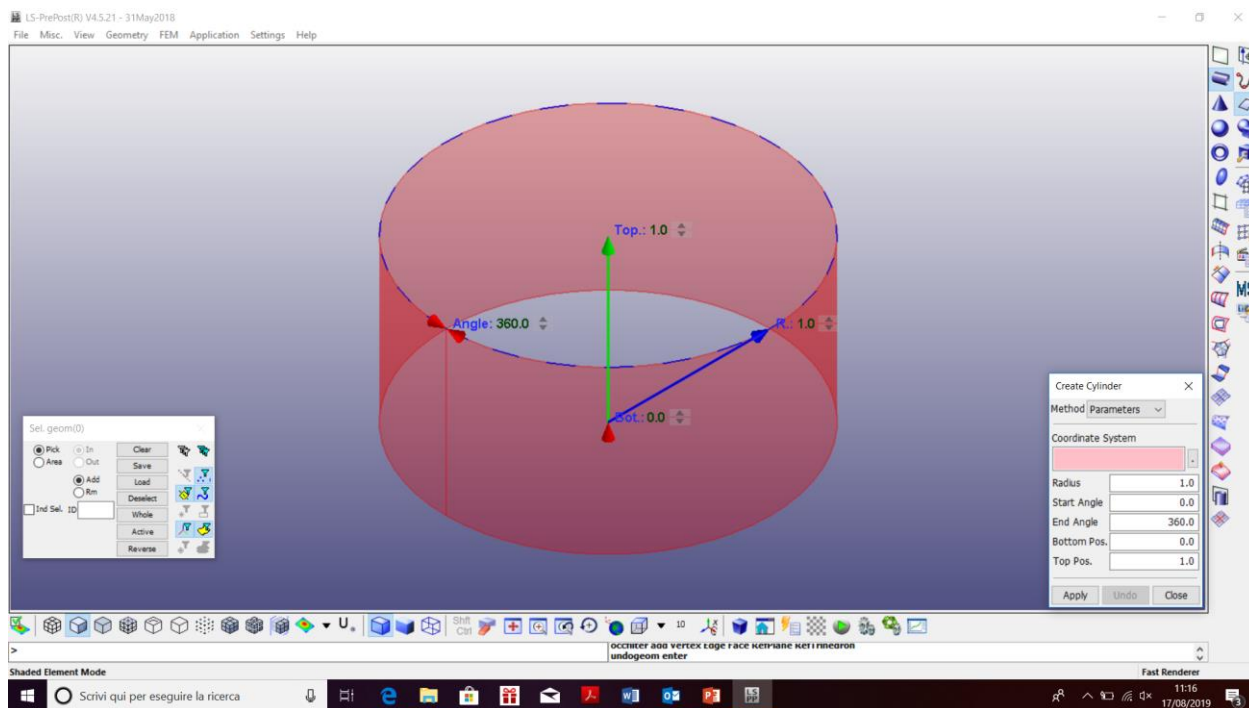
Definition of the geometry

Construction of the 3D model/models

Surface
modeling

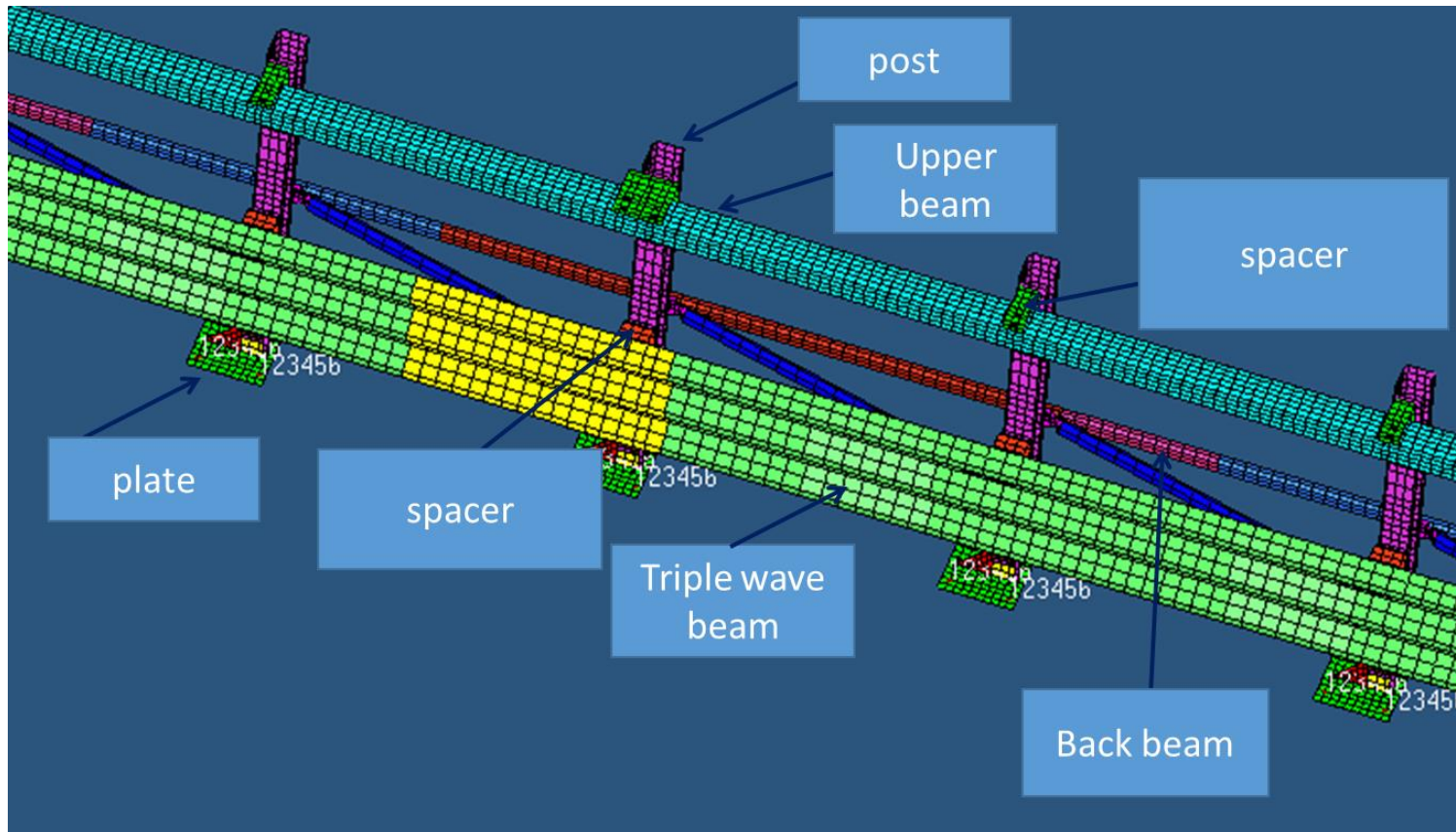


Mid surface



Pre-processing

FEM characterization



Hierarchical
approach

*part

*section

+

*mat

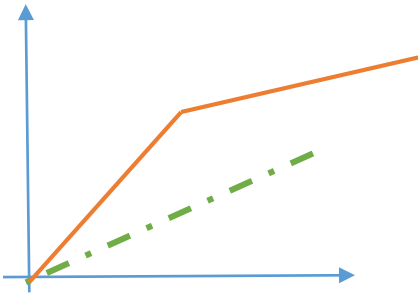
database

Pre-processing

FEM characterization

*mat

001 elastic



*MAT_24 (MAT_PIECEWISE_LINEAR_PLASTICITY)

LS-PrePost(R) V4.5.21 - 31May2018
 File Misc. Toggle Background Applications Settings Help

Keyword Input Form

NewID

RefBy

Add

Accept

Delete

Default

Done

☐ Use *PARAMETER
 (Subsys: 1)

Setting

*MAT_ELASTIC_(TITLE) (001) (0)

TITLE

1	MID	RO	E	PR	DA	DB	NOT USED
					0.0	0.0	0

COMMENT:

DA := Axial damping factor (used for Belytschko-Schwer beam, type 2, only).

Title	Off	Tims	Triad
Hide	Shad	View	Wire

*Airbag	*Dbase	*Mat
*Ale	*Define	*Node
*Boundry	*Elem	*Param
*Cnstrnd	*Eos	*Part
*Compnt	*Hrglass	*Rgdwal
*Contact	*Initial	*Section
*Control	*Intgrtn	*Set
*Def2Rg	*Intrfac	*Termnt
*Damping	*Load	*User

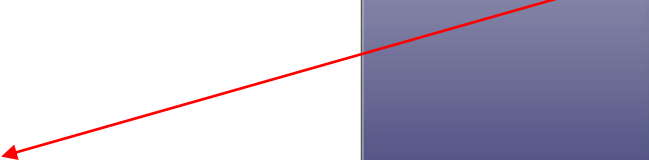
Keyword *MAT
 Transfer From MatDB
 Edit RefBy Done
 GroupBy Shell
 Sort Name All
 219-CODAM2
 022-COMPOSITE_DAMAGE
 118-COMPOSITE_DIRECT
 059-SHELL-COMPOSITE_FAILURE
 116-COMPOSITE_LAYUP
 117-COMPOSITE_MATRIX
 104-DAMAGE_1
 105-DAMAGE_2
 153-DAMAGE_3
 001-ELASTIC
 004-ELASTIC_PLASTIC_THERMAL
 106-ELASTIC_VISCOPLASTIC_THE
 060-ELASTIC_WITH_VISCOACITY
 060C-ELASTIC_WITH_VISCOACITY_I
 054/055-ENHANCED_COMPOSITI
 039-FLD_TRANSVERSELY_ANISO
 190-FLD_3-PARAMETER_BARLAT
 076-GENERAL_VISCOELASTIC
 254-GENERALIZED_PHASE_CHAN
 101-GEPLASTIC_SRATE_2000A
 280-GLASS
 120-GURSON
 120-GURSON_JC
 120-GURSON_RCDC
 243-HILL_90
 122-HILL_3R
 122-HILL_3R_3D
 203-HYSTERETIC_REINFORCFEMF

Pre-processing

FEM characterization

*section

Type of
element and #
of integration
point



LS-PrePost(R) V4.5.21 - 31May2018
 File Misc. Toggle Background Applications Settings Help

Keyword Input Form

NewID Draw RefBy Sort/T1 Add Accept Delete Default Done

☐ Use *PARAMETER (Subsys: 1) Setting

*SECTION_SHELL_TITLE (0)

TITLE

1	SECID	ELFORM	SHRF	NIP	PROPT	QR/IRID	ICOMP	SETYP
		2	1.0	2	1	0	0	1
2	I1	I2	I3	I4	NLOC	MAREA	IDOF	EDGSET
	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0

Repeated Data by Button and List

ELFORM := ELFORM Element formulation options:
 EQ.1: Hughes-Liu,
 EQ.2: Belytschko-Tsay (default),
 EQ.3: BCIZ triangular shell,

*Airbag	*Dbase	*Mat
*Ale	*Define	*Node
*Boundry	*Elem	*Param
*Cnstrnd	*Eos	*Part
*Compnt	*Hrglass	*Rgdwal
*Contact	*Initial	*Section
*Control	*Intgrtn	*Set
*Def2Rg	*Intrfac	*Termnt
*Damping	*Load	*User

1 2 3 4 5 6 7 D
 Keyword *SECTION
 Edit RefBy Done
 All Model

ALE1D
 ALE2D
 BEAM
 BEAM_AISC
 DISCRETE
 POINT_SOURCE
 POINT_SOURCE_MIXTURE
 SPRING_DAMPER
 SEATBELT
SHELL
 SHELL_EFG
 SHELL_THERMAL
 SHELL_XFEM
 SOLID
 SOLID_EFG
 SOLID_PERI
 SOLID_SPG
 SPH
 SPH_ELLIPSE
 TSHELL

Title Off Tims Triad
 Hide Shad View Wire

f11
 undogeom leave

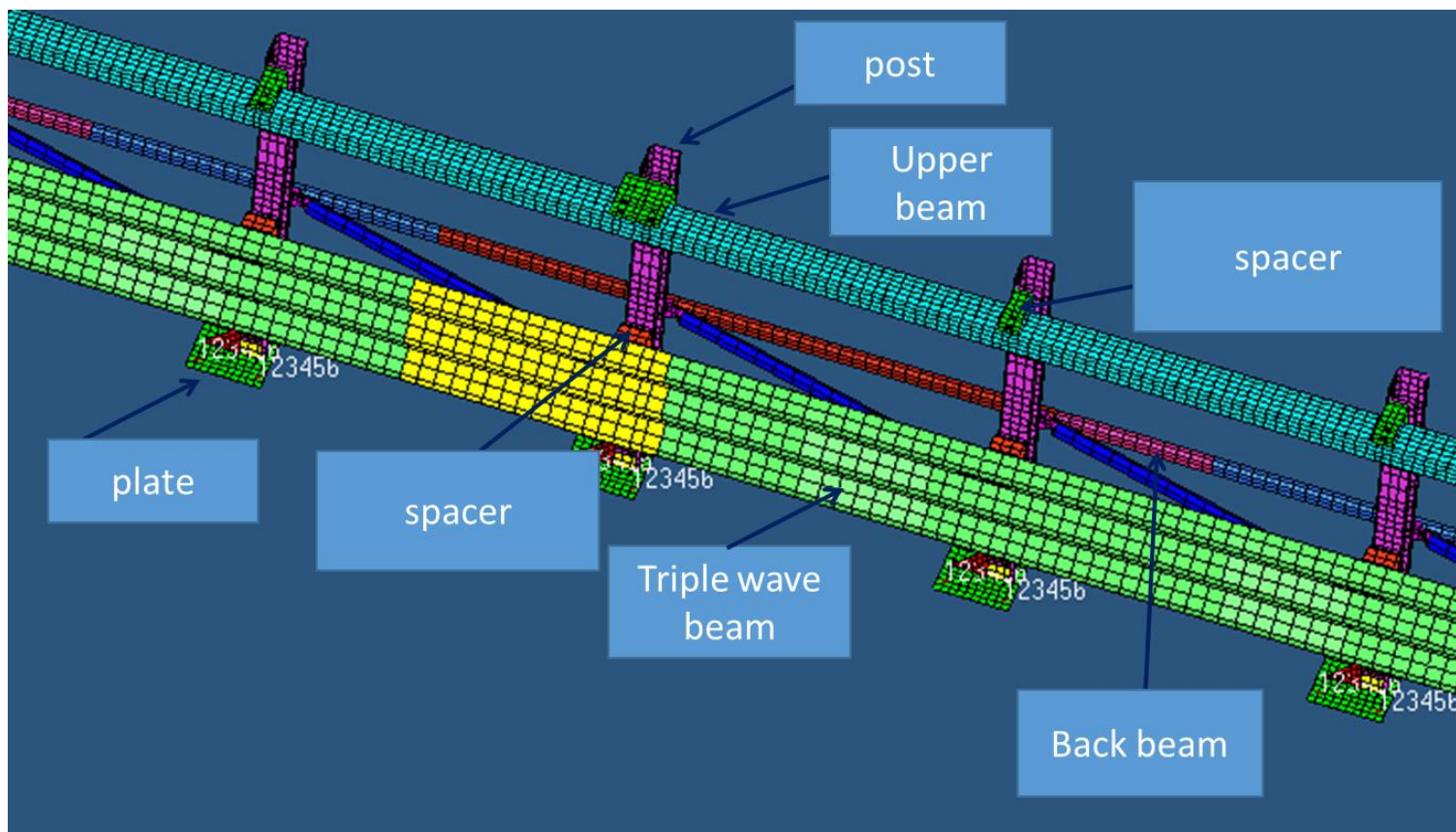
Enter data into text field

Scrvì qui per eseguire la ricerca

Fast Renderer
 11:33
 17/08/2019

Pre-processing

FEM characterization



*part

*section

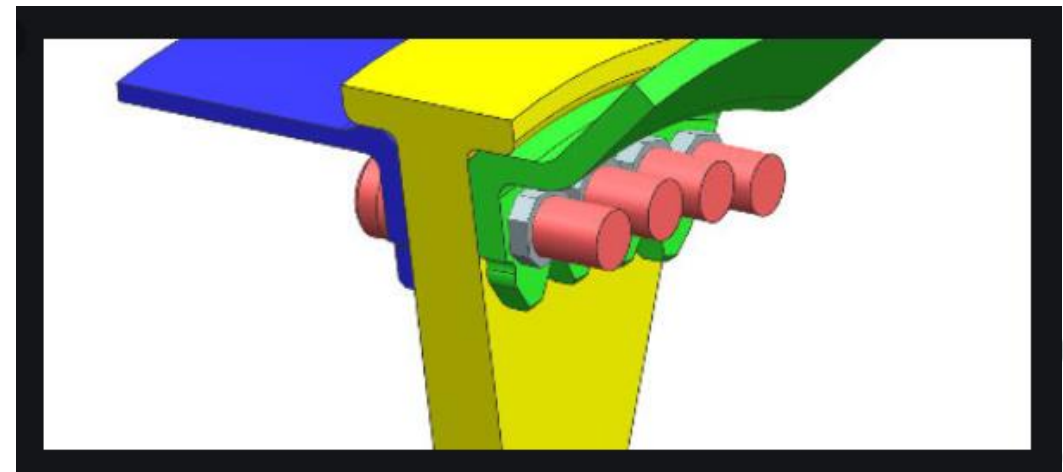
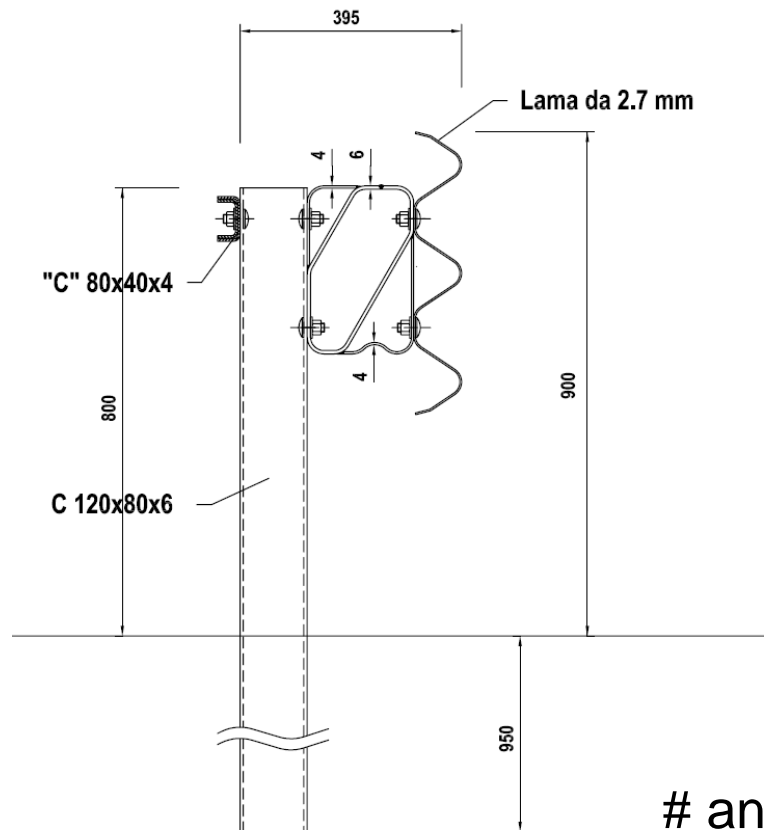
+

*mat

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

Pre-processing

FEM characterization



Connection between
different parts of the
model

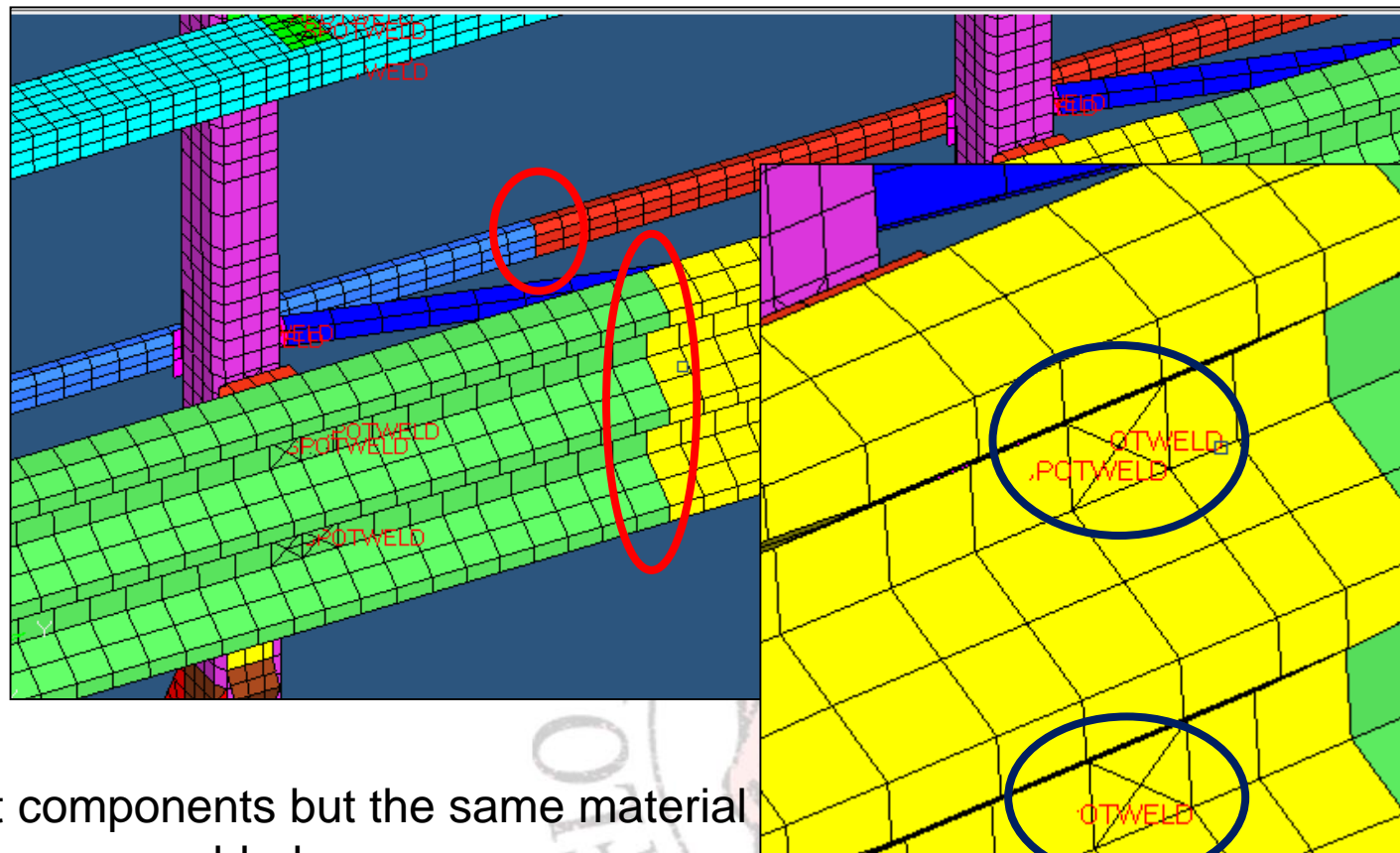
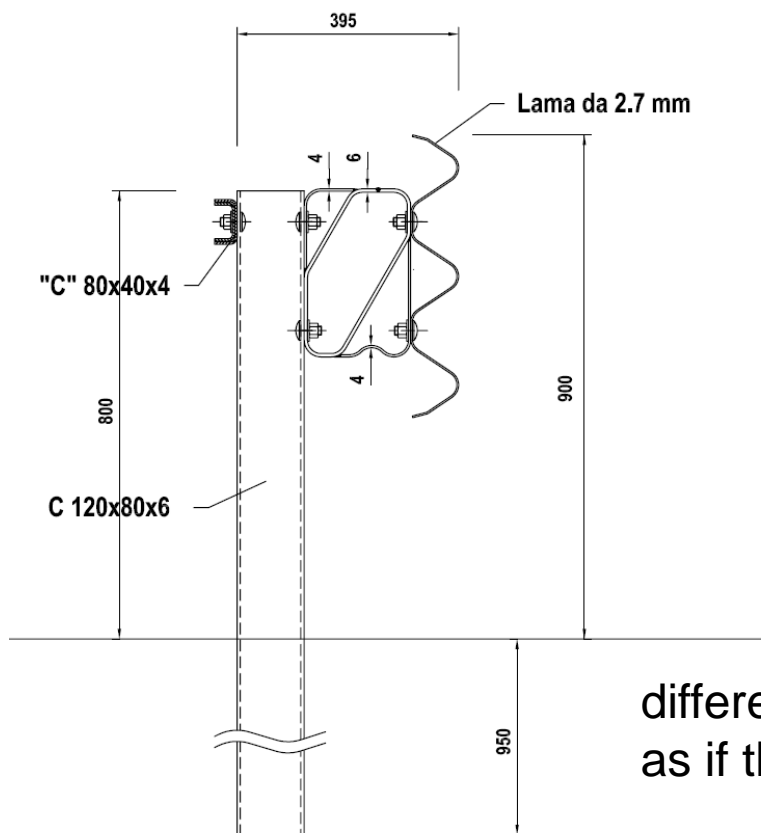


C/B analysis
will be
conducted

and dimensions of the elements → computational cost and findings

Pre-processing

FEM characterization

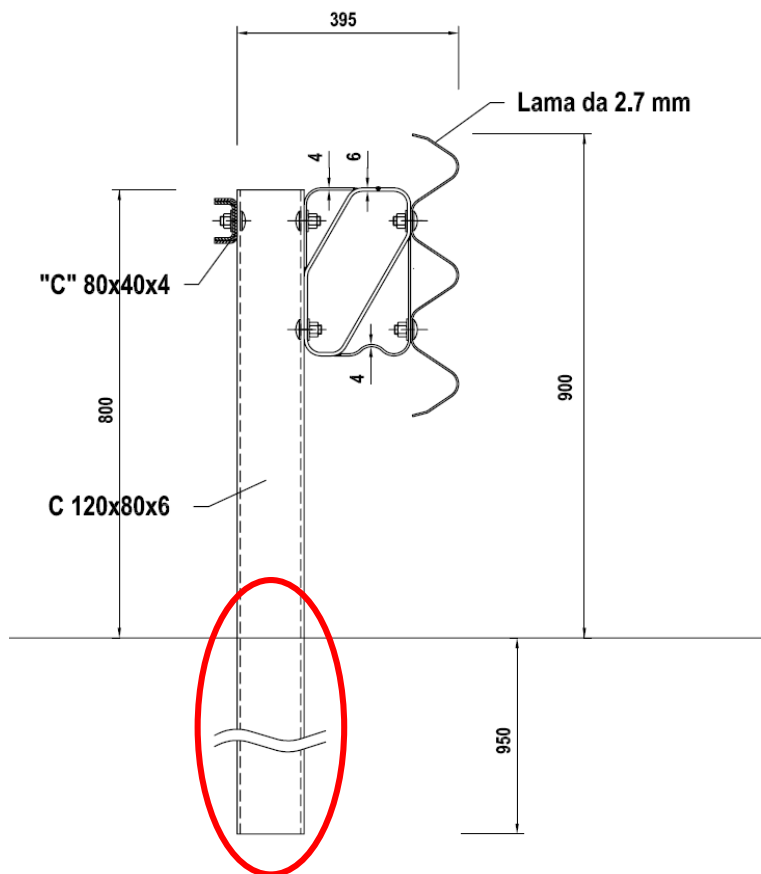


different components but the same material
as if they were welded

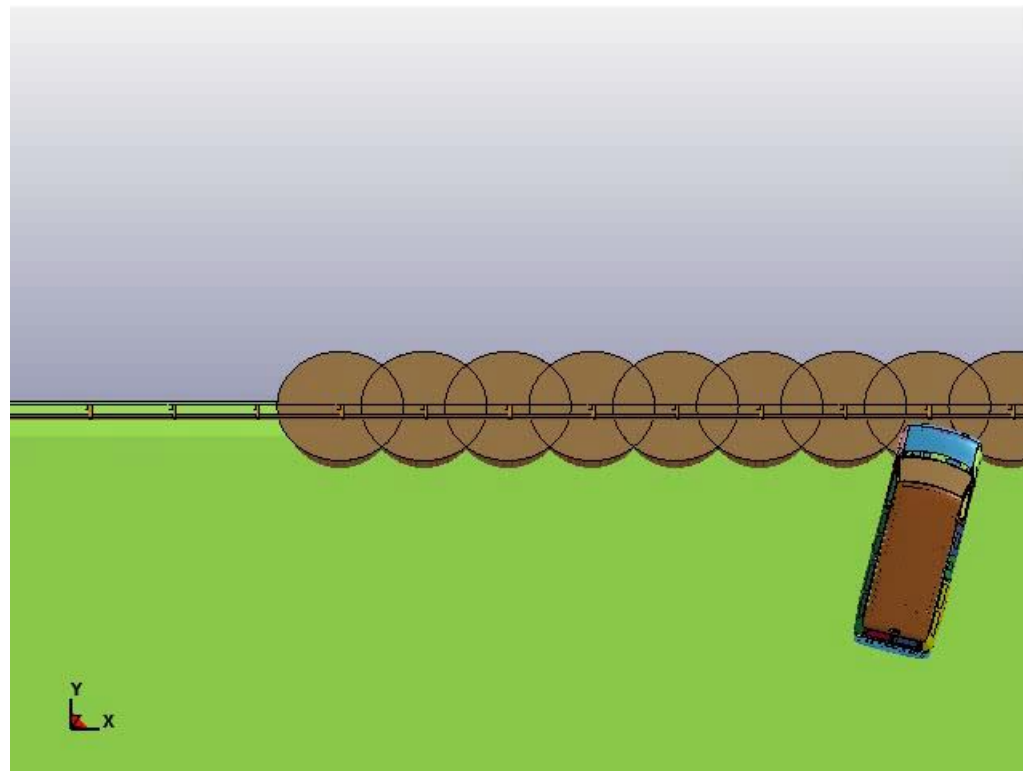
1-D elements → characterized by the same property
of the bolts

Pre-processing

FEM characterization



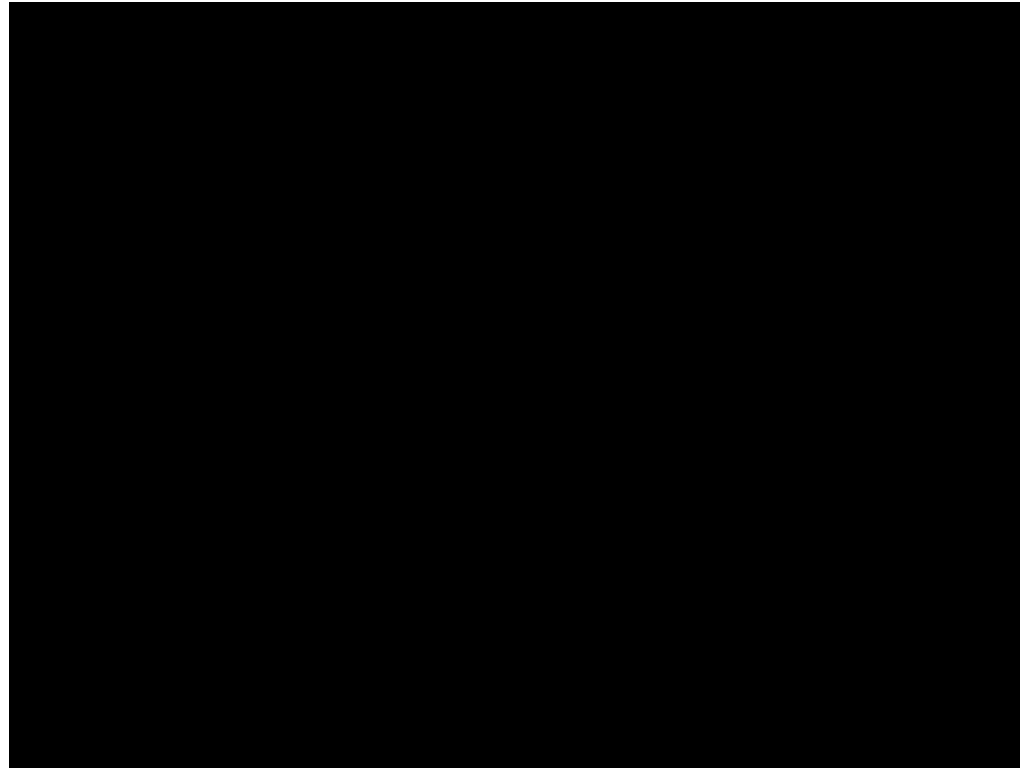
- 1) Soil modelling → solid element in order to reproduce the real effect



Pre-processing

FEM characterization

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

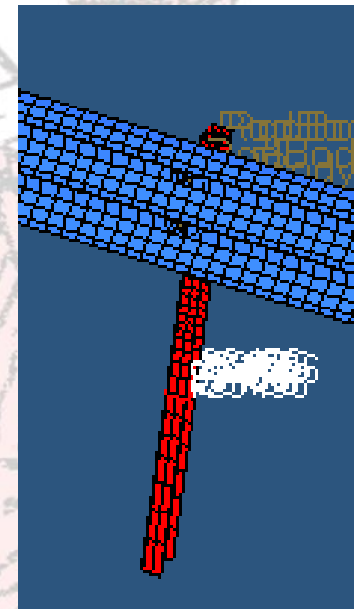
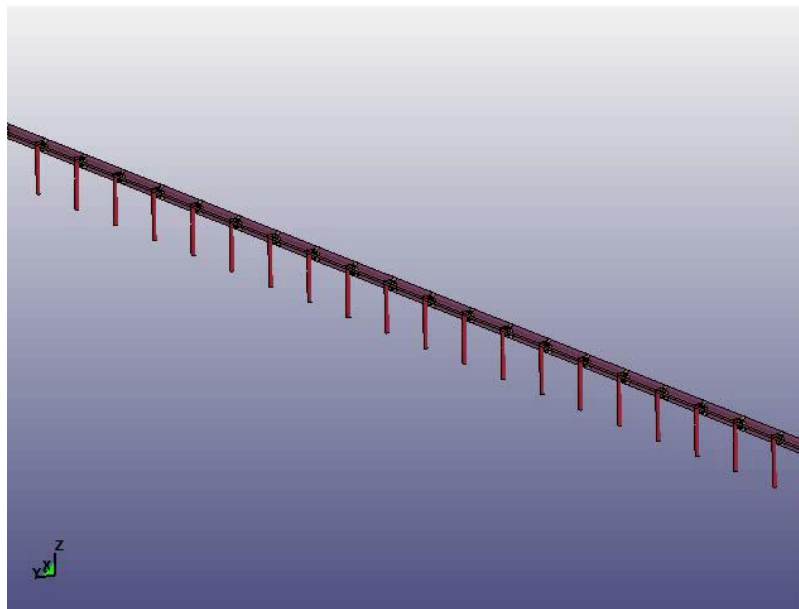
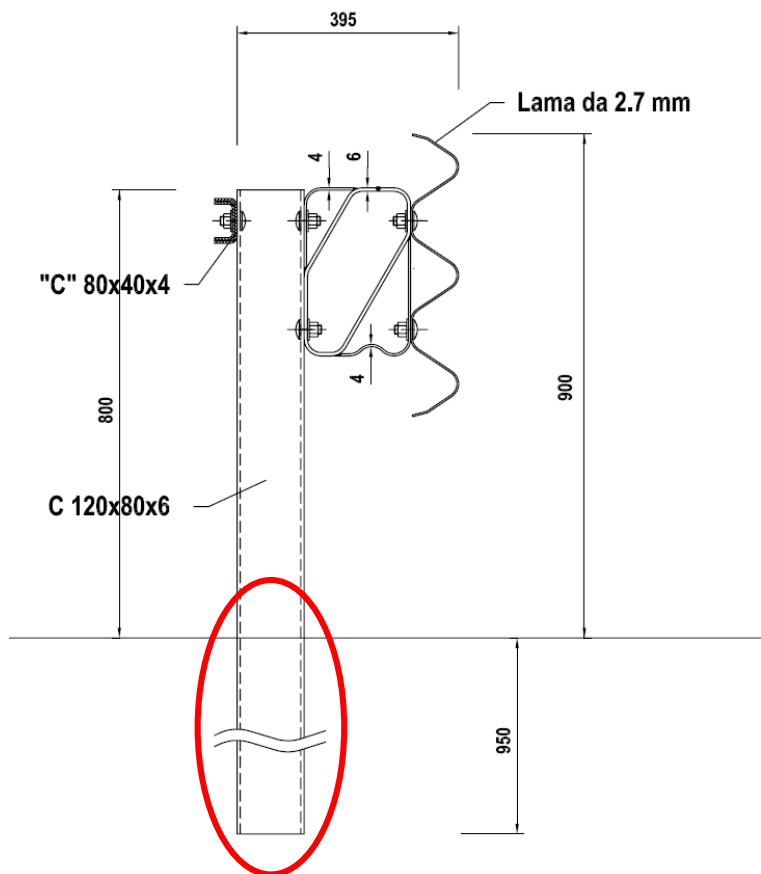


Pre-processing

FEM characterization

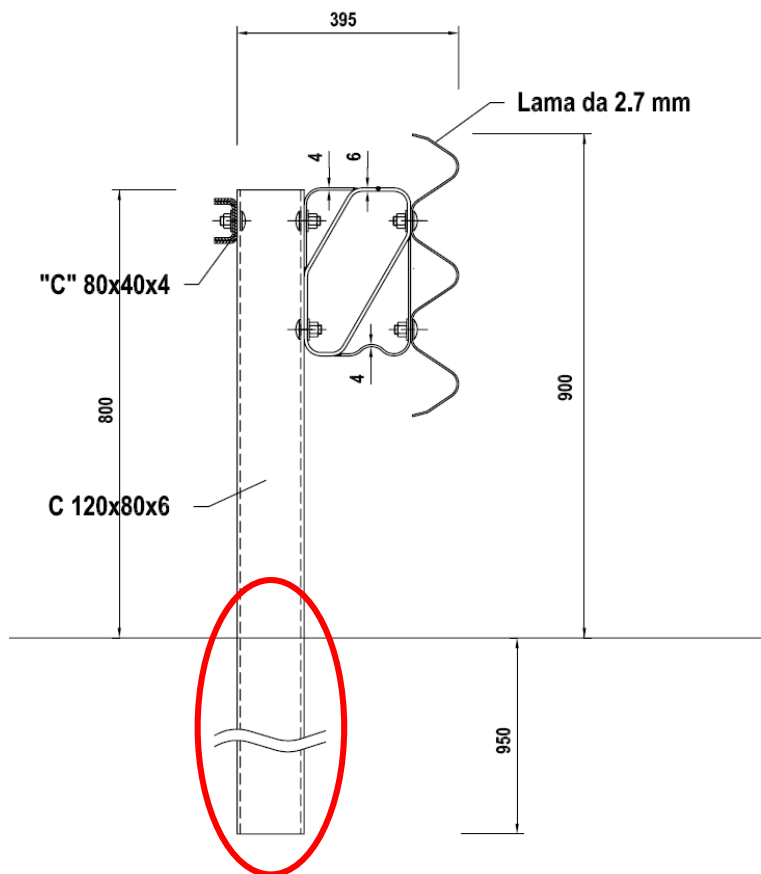
2) Definition of boundary condition

6 DoF \rightarrow x,y,z directions and 3 rotations



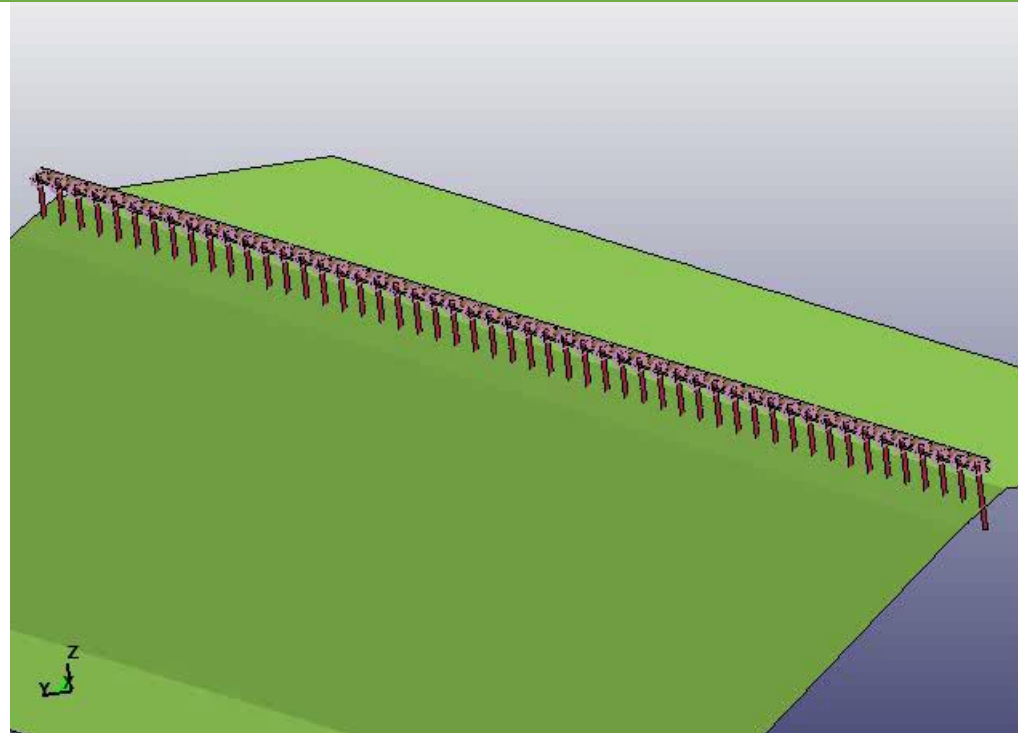
Pre-processing

FEM characterization



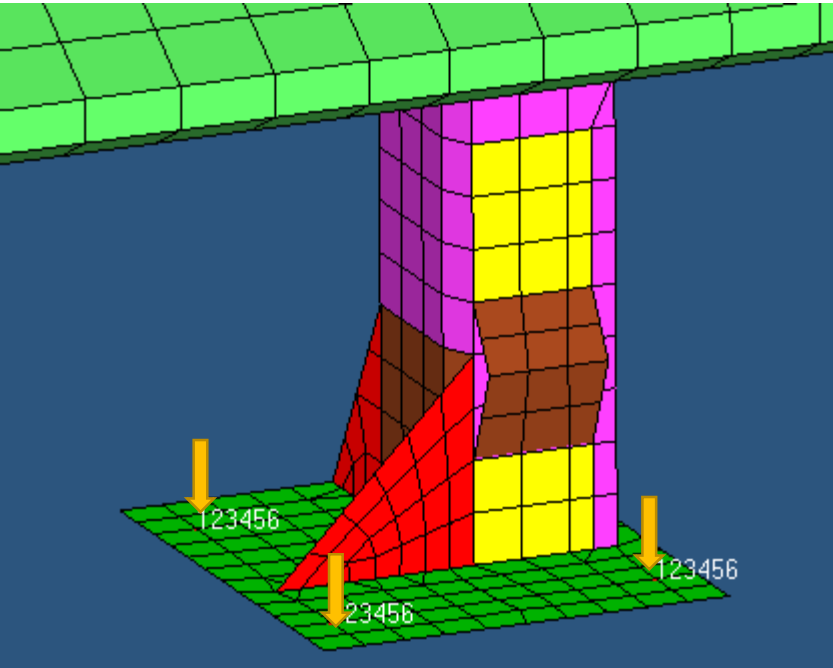
2) Definition of boundary condition

6 DoF \rightarrow x,y,z directions and 3 rotations

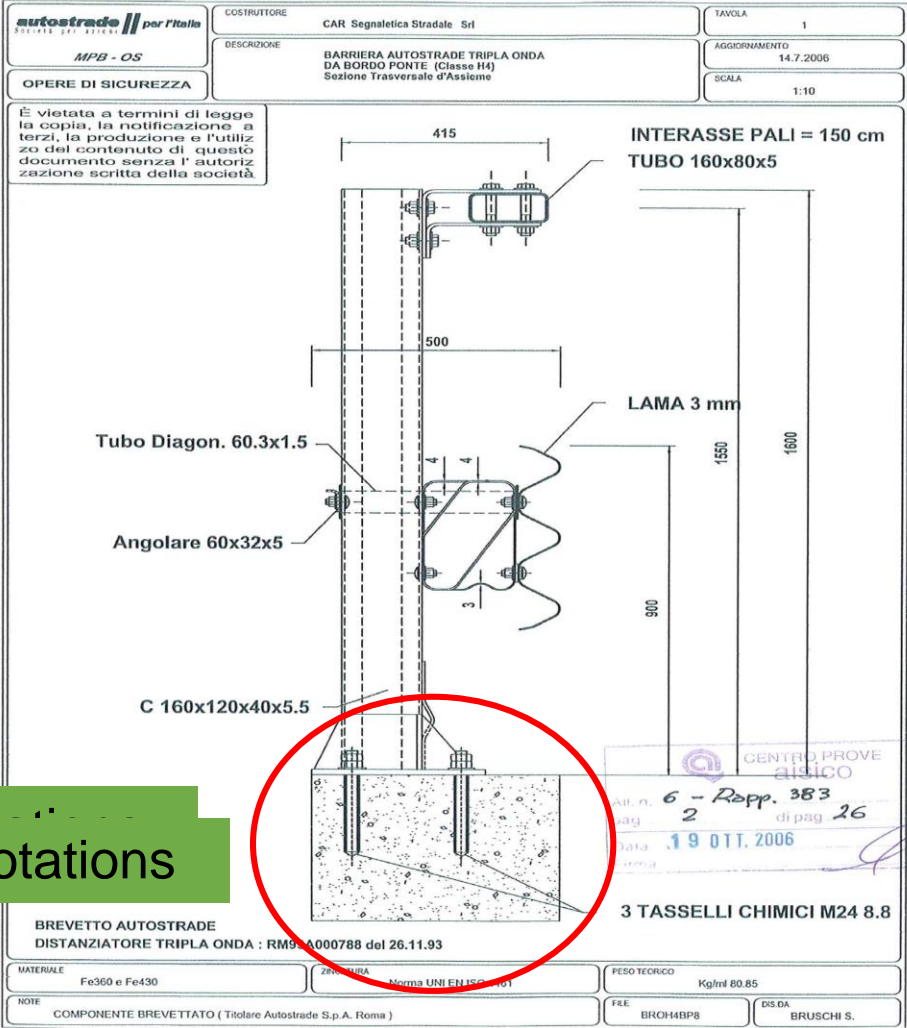


Pre-processing

FEM characterization

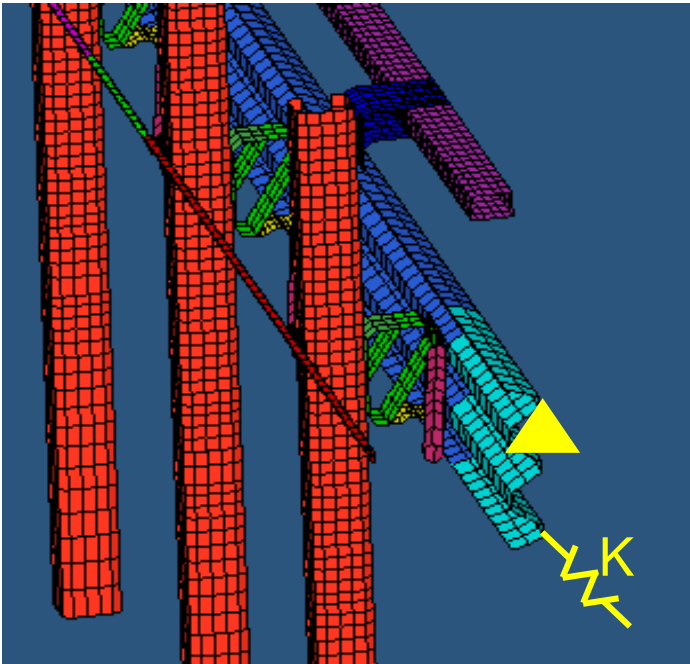


No translations/no rotations



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

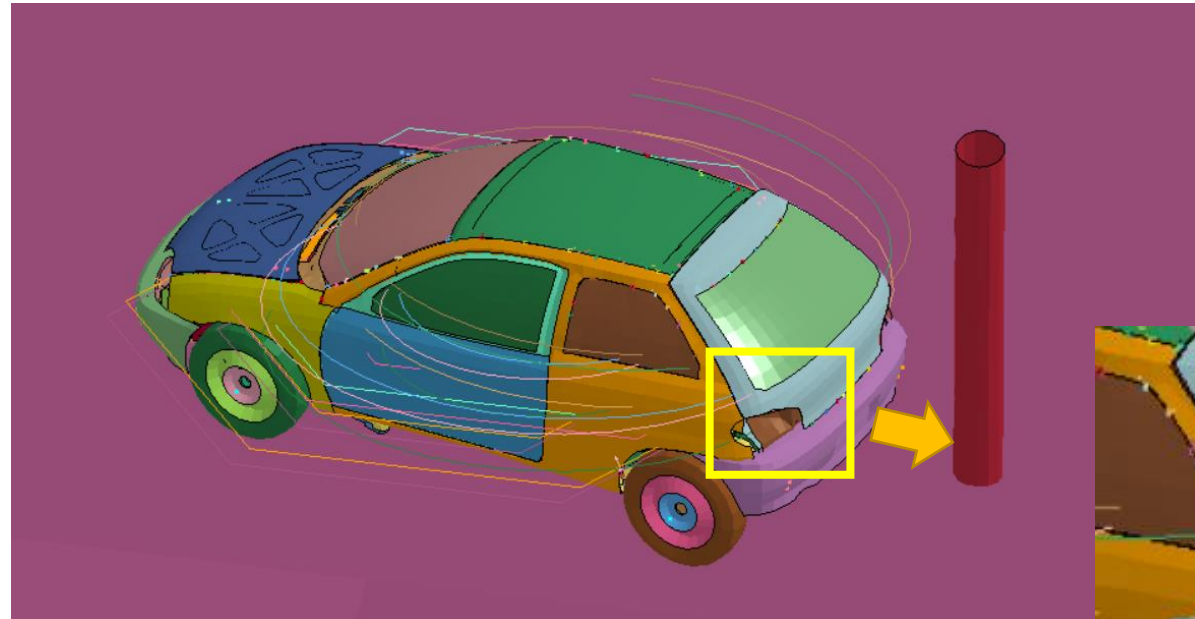
Contact → Interaction between two (or more) different object

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"



LS DYNA Software

Pre-processing

Contact → Interaction between two (or more) different object

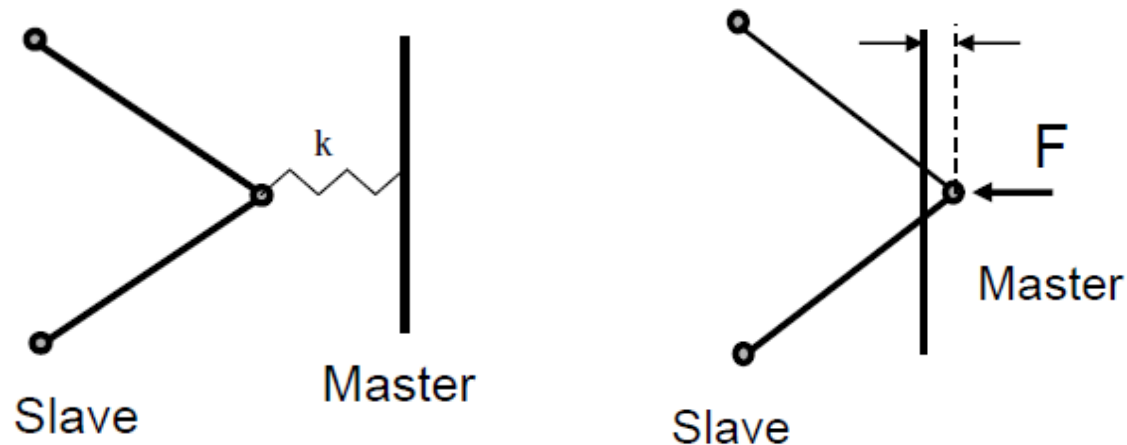
- Kinematic constraint Method;
- Penalty method;
- Distributed Parameter Method.



Pre-processing

Contact → Penalty methods

Main contact used for the reproduction of the crash phenomena



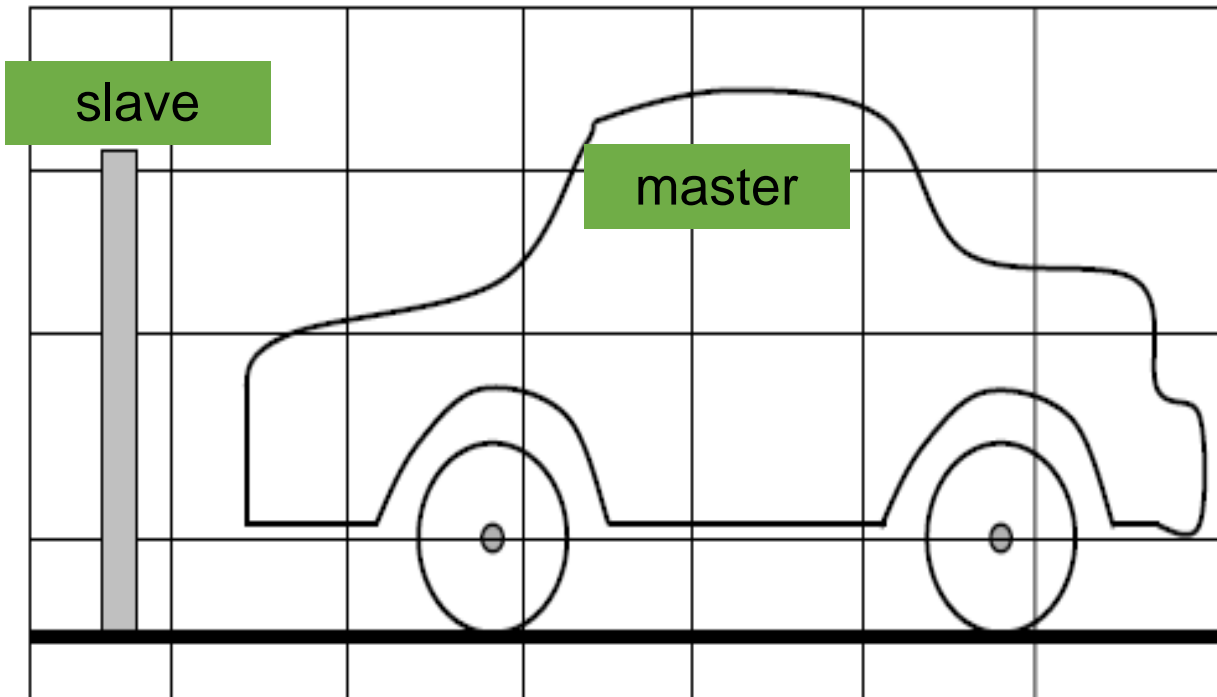
slave

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 constraint Method;
- Penalty method;
- Distributed 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 Constraint (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

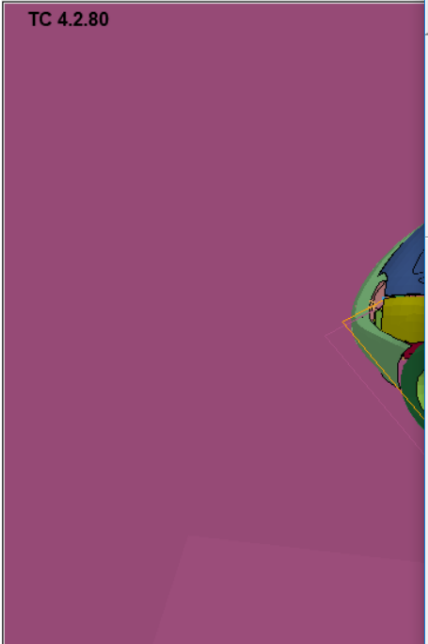
Contact

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-PrePost(R) V4.5.21 - 31May2018-64bit utente\Desktop\Monitor

File Misc. Toggle Background Applications Settings Help

TC 4.2.80



Triad Bcolr U

Wire Feat E

Keyword Input Form

NewID Draw

Pick Add Accept Delete Default Done

☐ Use *PARAMETER (Subsys: 1)

Setting

*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_ID/TITLE/MPP_(THERMAL) (31)

CID	TITLE
4000001	VEICOLO-STRADA

☐ MPP1 ☒ MPP2

IGNORE	BUCKET	LCBUCKET	NS2TRACK	INITITER	PARMAX	UNUSED	CPARM8
0	200	0	3	2	1.0005000	0	0

UNUSED	CHKSEGS	PENSE	GRPABLE
0	0	1.0000000	0

SSID	MSID	SSTYP	MSTYP	BOXID	MBOXID	SPR	MPR
35000000	421	3	2	0	0	0	0

FS	FD	PL	VC	VDC	PENCHK	BT	DT
0.6000000	0.7000000	0.0	0.0	0.0	0	0.0	1.000e+20

SFS	SFM	SST	MST	SFST	SFMT	SFS	VSF
1.0000000	1.0000000	60.000000	60.000000	1.0000000	1.0000000	1.0000000	1.0000000

☐ Thermal ☐ T_Friction ☐ A ☐ AB ☐ ABC ☐ ABCD

Total Card: 31 Smallest ID: 1 Largest ID: 35 Total deleted card: 0

1 (4000001) VEICOLO-STRADA

2 (4000003) VEICOLO-MANICHII

3 (4000004) VEICOLO-MANICHII

4 (4000005) MANICHINO1-MAN

5 (4000003) VEICOLO-ALBERO

6 (20000001) PPP

9 (20000001) PPP

10 (20000001) PPP

11 (20000001) PPP

12 (20000001) PPP

13 (20000001) PPP

14 (20000001) PPP

15 (20000001) PPP

16 (20000001) PPP

17 (20000001) PPP

18 (20000001) PPP

19 (20000001) PPP

20 (20000001) PPP

21 (10000001) PPP

24 (10000001) PPP

25 (10000001) PPP

26 (10000001) PPP

27 (10000001) PPP

28 (10000001) PPP

29 (10000001) PPP

30 (10000001) PPP

31 (10000001) PPP

32 (10000001) PPP

33 (10000001) PPP

34 (10000001) PPP

35 (10000001) PPP

*Airbag *Dbase *Mat

*Ale *Define *Node

*Boundry *Elem *Param

*Cnstrnd *Eos *Part

*Compnt *Hrglass *Rgdwal

*Contact *Initial *Section

*Control *Intgrtn *Set

*Def2Rg *Intrfac *Termnt

*Damping *Load *User

Keyword *CONTACT

Edit RefBy Done

All Model

[*]AUTOMATIC_NODES_TO_SURFACE
 [*]AUTOMATIC_SINGLE_SURFACE
 [*]AUTOMATIC_SURFACE_TO_SURFACE
 [*]TIED_SURFACE_TO_SURFACE (1)

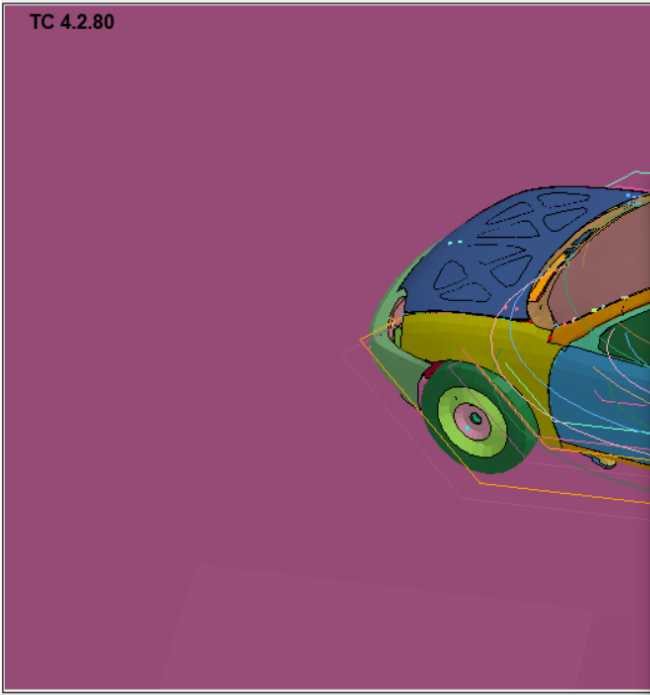
quat 0.522500 0.013500 0.076254 0.849115;
 zoomhere 0.089474 0.204340 0.028147;

Pre-processing

*Initial ...velocity

LS-PrePost(R) V4.5.21 - 31May2018-64bit utente\Desktop\Monica\perizie con simulazioni\terni\Main.dyn
 File Misc. Toggle Background Applications Settings Help

TC 4.2.80



Title	Off	Tims	Triad	Bcolr	Unode	Frin	Isos
Hide	Shad	View	Wire	Feat	Edge	Grid	Mesh

Keyword Input Form

NewID

Draw

Pick

Add

Accept

Delete

Default

Done

☐ Use *PARAMETER

(Subsys: 2)

Setting

*INITIAL_VELOCITY_GENERATION (3)

	NSID/PID	STYP	OMEGA	VX	VY	VZ	IVATN	ICID
1	20000179	3	0.0	1.305e+04	-4750.2808	0.0	0	0
	XC	YC	ZC	NX	NY	NZ	PHASE	IRIGID
2	1799.9999	-3240.0000	-1100.0000	0.0	0.0	1.0000000	0	0

COMMENT:

Total Card: 3 Smallest ID: 1 Largest ID: 3 Total deleted card: 0

*Airbag	*Dbase	*Mat
*Ale	*Define	*Node
*Boundry	*Elem	*Param
*Cnstrnd	*Eos	*Part
*Compnt	*Hrglass	*Rgdwal
*Contact	*Initial	*Section
	Intgrtn	*Set
	Intrfac	*Termnt
	*Load	*User

RefBy

Done

Model

GENERATION (3)

quat 0.522500 0.013500 0.076254 0.849115;
 zoomhere 0.089474 0.204340 0.028147;

*Define curve...



UNIVERSITÀ
DEGLI STUDI
FIRENZE
DICEA
DIPARTIMENTO
DI INGEGNERIA CIVILE
E AMBIENTALE

LS DYNA Software

...and practically....

