

GiD-LsDyna problemtype

Interface with LsDyna

- 1. GiD-LsDyna tutorial 2
- 1.1 LsDyna tutorial 2

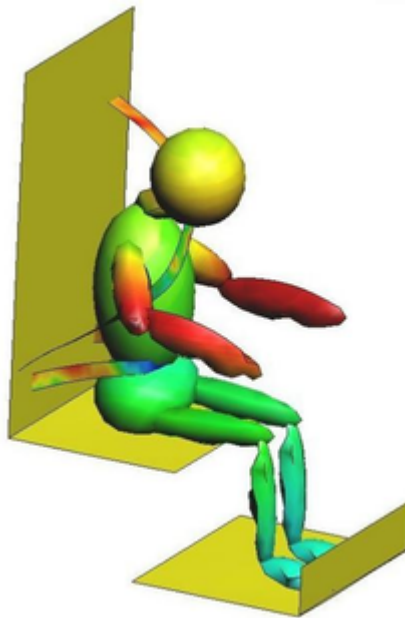
GiD-LSDyna tutorial

LsDyna tutorial

SIMPLE CRASH TEST PRE AND POST-PROCESSING

The objective of this case study is to pre and post-processing a simple LS-DYNA Crash test. The analysis is carried out in nine steps:

- GiD-LSDYNA installation
- General data definition
- Part definition
- Constraints definition
- Initial conditions definition
- Postprocessing options selection
- Model Meshing
- Simulation run
- Results viewing



GiD-LSDYNA INSTALLATION

At the moment, GiD-LSDYNA interface is only available in Windows OS. Follow these instructions to install it:

1. Install GiD Software on your computer
2. Install Lsdyna interface from Internet Retrieve section (Data menu). Lsdyna will be ticked in available problemtypes list (**Data->Problem type** of GiD menu). If not, please select it.
3. By default, the interface menu (a data tree showing created groups and problem data) should appear on the left of your screen. If you can't see it, please click (**Data->Data(internal)** of GiD menu).

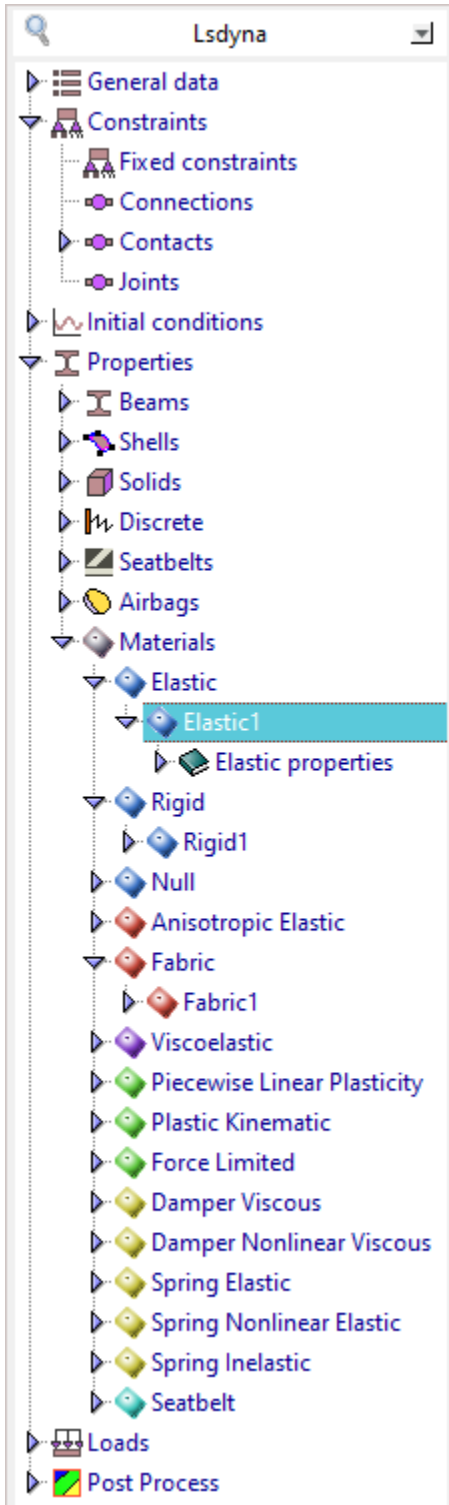
GENERAL DATA DEFINITION

You can also find the model of this case in the Internet Retrieve section. It should be installed in your GiD Examples folder (for instance <GiD>/Examples/Lsdyna). The model is named *gid_lsdyna_tutorial.gid*.

- Open our case. **Files->Open** of GiD menu and select *gid_lsdyna_tutorial.gid* path
- Choose **Render->Flat** from the mouse menu if this option is not selected (right click on screen)
- Choose **Rotate->Trackball** from the mouse menu (this tool is also available by pressing ALT key+ left click). Make a few changes to the perspective in order to get an idea of the geometry under study
- Edit **General Data** information in the interface menu:
 - In **Units** menu, set **Mesh Units** to cm (to represent human being sizes)
 - In **Gravity** menu, activate gravity and set its value to $9,81 \text{ m/s}^2$ (in X+ sense)

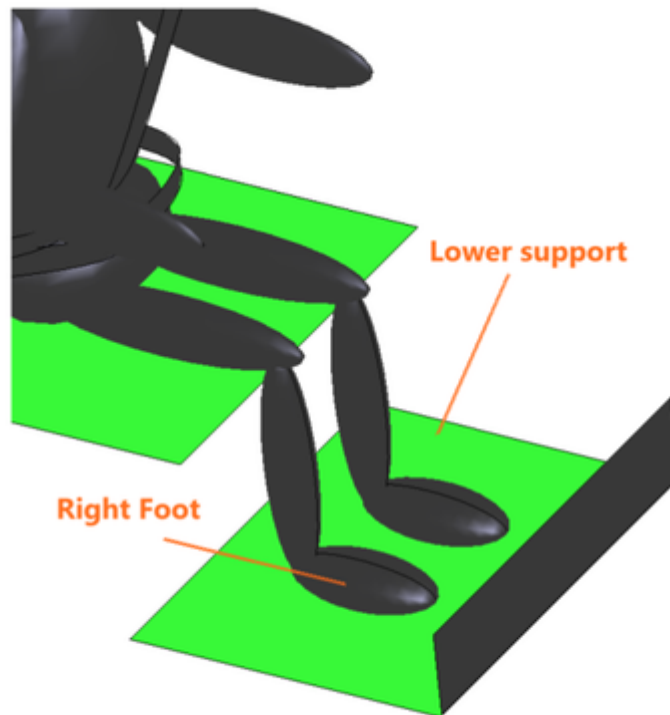
PART DEFINITION

- GiD-LS DYNA Interface is based in *group* concept. A *group* is a group of geometrical entities or mesh elements where one or several conditions are applied. These conditions could define geometrical properties and materials, loads, constraints, etc. Property groups are very important in this case, because in LS-DYNA every element must be related with a material (it means that in our interface each geometrical entity should be related with a single property group).
- Most of problem parts are already defined as groups. You can notice it by drawing shell parts. Please right click **Properties->Shells** in the interface menu. Select **Draw-Groups** option.
- First we must define materials. By default, one material sample of each type has been created. More materials could be created (right click in a material type) and modified with the interface.




To modify material properties user just have to double click material's name and edit each field like another interface group. In this case we will use *Elastic1* and *Rigid1* materials, but we don't need to modify its properties (we will use default values).

- To define Lower Support shell:
 - Double Click in **Shells->Isotropic Shell**
 - Set **Thickness** to 0.01 m
 - Select **Elastic1** Material in material drop box
 - Click in Group command (icon placed next to drop list) and select Lower support surface form the screen. Press ESC after selection.
 - Change group name to *Lower Support* (optional)
 - Click OK to create this part
 - To define Right foot:
 - Double Click in Shells->Isotropic Shell
 - Set **Thickness** to 0.01 m
 - Select **Rigid1** Material in material drop box
 - Click in Group command (icon placed next to drop list) and select Right Foot surface form the screen, if render problems appears please rotate the model to solve them. Press ESC after selection.
 - Change group name to *Right Foot* (optional)
 - Click OK to create this part
-



CONSTRAINTS DEFINITION

- To represent the crash seat will suffer an important deceleration. To apply this constraint please follow these steps:
 - Double Click in **Constraints->Fixed Constraints**
 - In *Fixation* tab, only tick **Z Constraint** option (acceleration will be prescribed in Z global axes direction)
 - Select *Acceleration* tab (left click)
 - Click  to edit the acceleration behavior
 - In data entry options, verify that *t units* are set to s
 - Choose *linear ramp* to edit four different ramps with the parameters below

(click *Add at the end* after each curve data introduction):

Curve part 1:

- *t0*: 0
- *tend*: 0.001

- *Factor0*: 0
- *Factorend*: 0
- *t*: 0.0001

Curve part 2:

- *t0*: 0.002
- *tend*: 0.004
- *Factor0*: 0
- *Factorend*: 1
- *t*: 0.0001

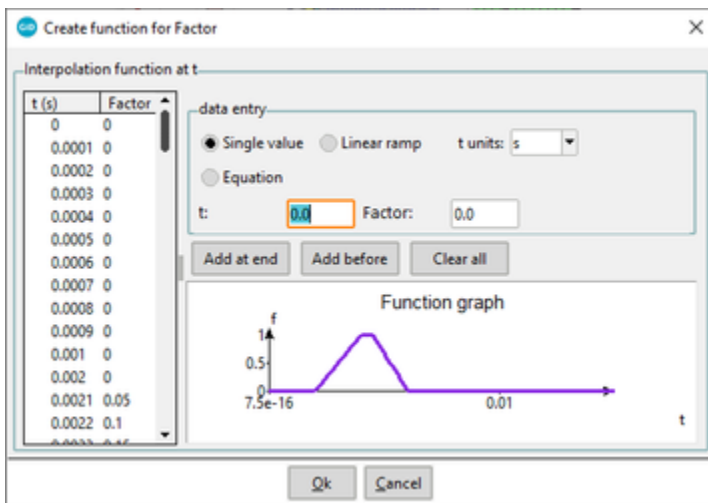
Curve part 3:


- *t0*: 0.0045
- *tend*: 0.006
- *Factor0*: 1
- *Factorend*: 0
- *t*: 0.0001

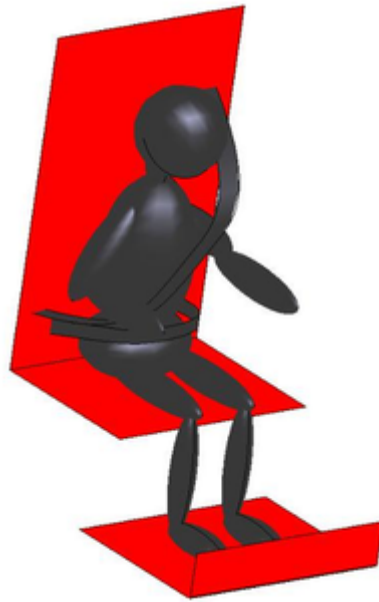
Curve part 4:


- *t0*: 0.007
- *tend*: 0.015
- *Factor0*: 0
- *Factorend*: 0
- *t*: 0.002

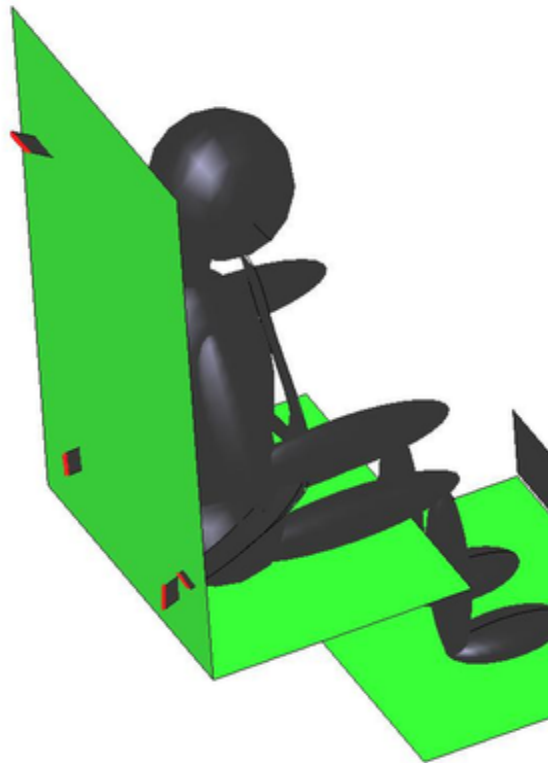
- Click *Ok* to close function editor





- Set *az* value to -15000 m/s^2 . This factor will multiply the defined curve
- Select surface icon  from the available entities.
- Click group command and select Seat surfaces. Press ESC after selection

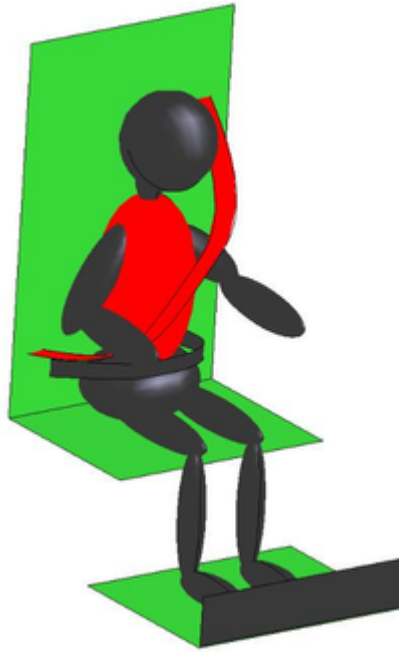


- Name this group 'SeatBelt acceleration' (optional)
- To include seatbelt to this prescribed motion, please go to Data->Groups in GiD menu and select the group created in the last step from the list. Click *Assign entities to group*  to add seatbelt fixation surfaces, like the image. Press ESC after selection. Click *finish* and close groups editor.

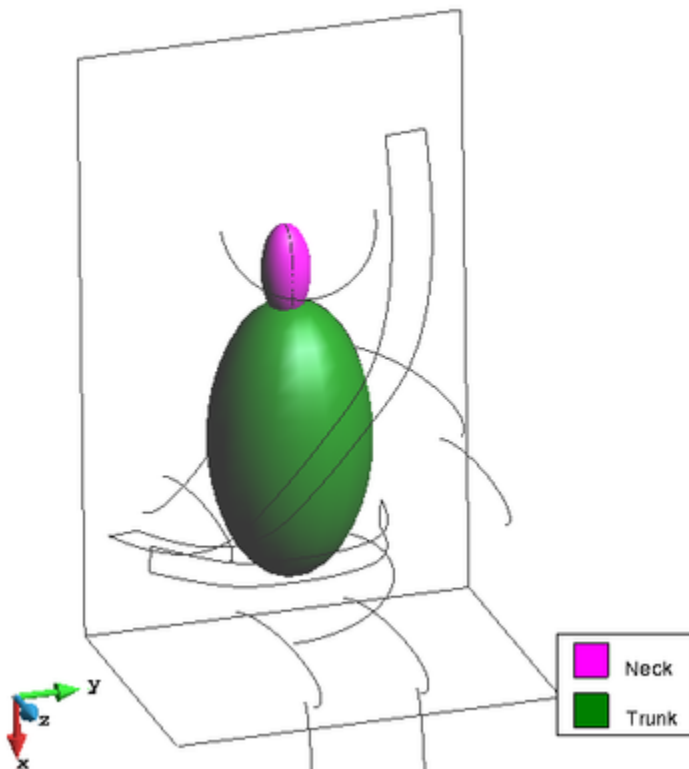




- Most part of contacts are already defined. The interface supports several single and master slave contacts, but in this case only *Surface to Surface* contact type is applied. To prescribe contact between Trunk and the Seat belt (brest) please follow next instructions:
 - In **Constraints->Contacts->Control** set orientation to *Part and segment input*
 - Double click **Constraints->Contacts->Master Slave**
 - Select  from the available geometry entities
 - In *Slave*, select Seat belt breast surface with the group edition command  **Select**

- In *Master*, select Trunk surface with the group edition command



- Click OK to finish contact definition
- Most part of joints are already defined. With GiD-LSDYNA interface user can create spherical, revolute and translational joints, and define stiffness parameters.
- To define the joint between neck and trunk:
- Note: The stating model has already defined the groups 'Neck' and 'Trunk' and defined shell parts with these groups. In case that your model doesn't has this information, must define with (otherwise can skip this step)
 - Create first with **Utilities->Layers and groups** a group named 'Neck' and assign to this group the surfaces of the neck, and another group named 'Trunk' and assign the surfaces of the trunk



- and to define the part do it like in previous step, when defining the 'Right foot': (in **Shells->Isotropic Shell** , set **Thickness** to 0.01 m, **Rigid** material, and the existing group 'Neck', and repeat it for the 'Trunk'
- Then can define your joints
 - Double Click in **Constraints->Joints**
 - Set Joint Type to *Spherical and* Relative Penalty to 0.0
 - In *Point 1* tab, introduce P1 x= - 54 m, P1 y=0, P2 z=0.2 m. Points can also be selected from the screen with: 
 - Click *Draw Joint Points* button (placed under point coordinate labels) to draw entered point (after group creation, please press ESC on the screen to stop joint points viewing)
 - Select  from the available geometry entities
 - In *First Entity*, select Neck from the drop list
 - In *Second Entity*, select Trunk from the drop list
 - Press Ok

INITIAL CONDITIONS DEFINITION

In **Initial Conditions->All**, set velocity z to 56 m/s (this initial velocity will affect all nodes)

POSTPROCESSING OPTIONS


- Double click in **Post Process** to edit GiD postprocessing options
- Set interval to 0.001 s (at the bottom of the window)
- Tick **Write displacements**, **Write velocities** and **Write accelerations** (Nodal)
- Tick **Write shell stresses** and **Write shell strains** (Shells)

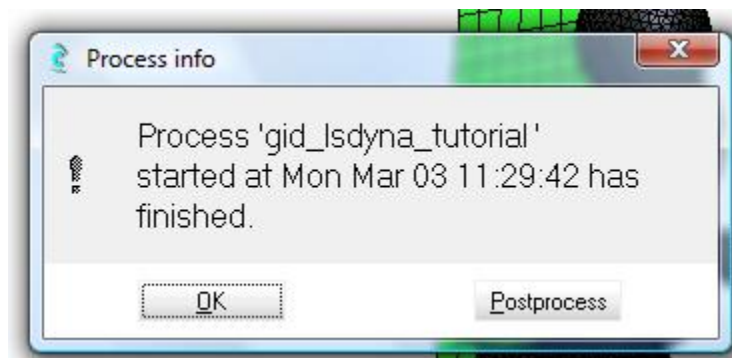
MODEL MESHING

There many meshing options implemented in GiD software. However, in this tutorial mesh creation have been simplified as much as possible. If you want to take contact with GiD meshing capabilities please visit GiD tutorials


- Choose **Render->Normal** from the mouse menu (right click)
- From GiD Menu, select **Mesh->Structured->Surfaces->Assign number of divisions to surface lines** and select seat surfaces. Press ESC to finish.
- Set 15 as number of cells to assign to surfaces and select seat lines in Y direction (lines will be in red). Press ESC to finish.
- Set 10 as number of cells to assign to surfaces and select seat lines in Z direction. Press ESC to finish.
- Set 20 as number of cells to assign to surfaces and select seat lines in X direction. Press ESC twice to finish.
- Choose **Render->Flat** from the mouse menu (right click on screen)
- From GiD Menu, select **Mesh->Element type->Quadrilateral**. Select Seat surfaces. Press ESC to finish.
- From GiD Menu, select **Mesh->Generate Mesh** and set 2 as mesh size.

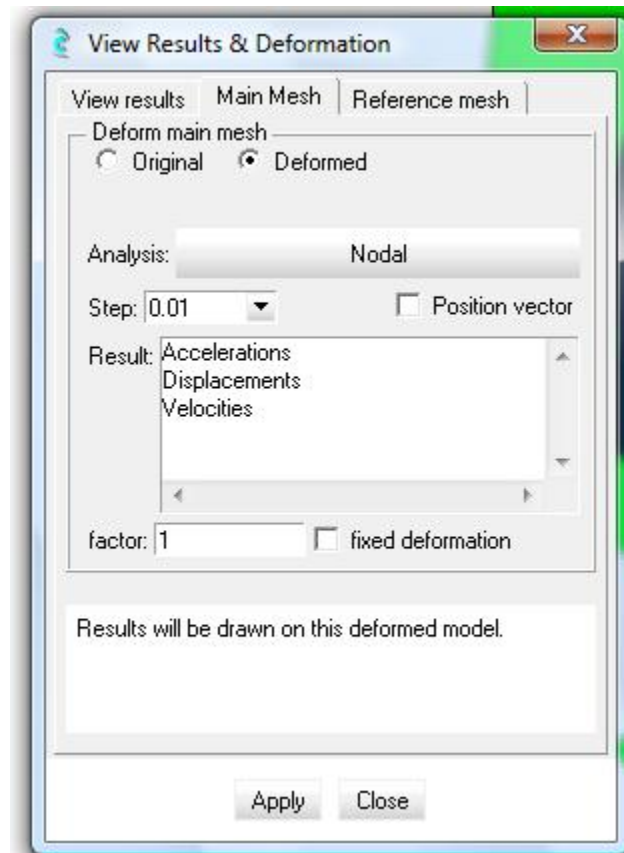
SIMULATION RUN

- From GiD Menu, select **Calculate->Solver Path** to choose your correct solver path (it will be saved for future simulations).
- To run the simulation, select **CalculateCalculate**
- 3. With the button  (placed in left toolbar) user can control simulation in real time
- 4. Wait until finish message appears

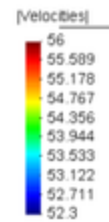
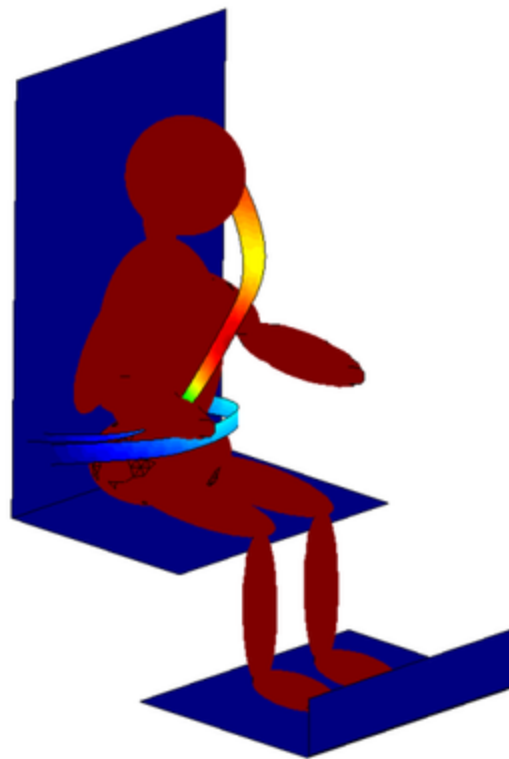
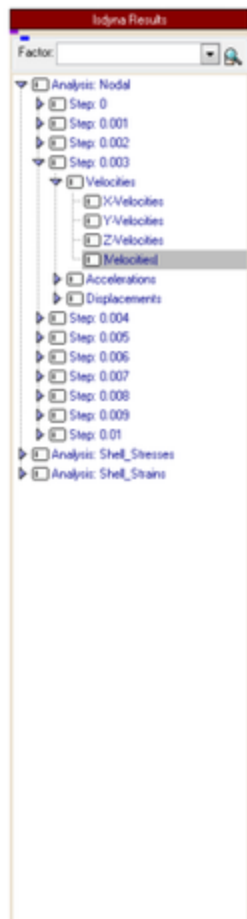


RESULTS VIEW

- When project finishes, a window appears allowing the user to analyze simulation results (click *Postprocessing* button). Postprocessing results are also available by  click in toolbar icon.
- After results load, a postprocessing interface menu will appear on the left of the screen
- To animate the results:
 - Select **Window->View Results** from the GiD menu
 - In Main Mesh Flap, tick *Deformed* and select *nodal* as Analysis type and *Displacements* as results .
 - Introduce 1 in Factor Field
 - Click *Apply* and *Close*
 - Select **Window->Animate...** from the GiD menu
 - Click *Play* Button



- From the postprocessing interface menu, select Analysis: **Nodal->Step:0.003->Velocities->|Velocities|** to study velocity contours at t=0.003 s



Nodal, step 0.003
Contour Fill of Velocities, [Velocities].