Application: AEF-A.17
Dynamic analysis of collision

KEY WORDS
Dynamic Analysis, Plane Geometric Model, Plane Stress State, Linear Material, 1D Finite Element, Linear Finite Element, Machine Element, Collision

CONTENT

A. PROBLEM DESCRIPTION

A.1 Introduction

The Ansys Workbench E*xplicit dynamics* suite it enables to capture the physics of shortduration events for products that undergo highly nonlinear, transient dynamic forces. In many cases, the accuracy of an explicit solution can be verified only via comparison with physical experiments. For some problems (such as explosions), it may be too expensive or impossible to perform tests.

"Implicit" and "Explicit" refer to two types of time integration methods used to perform dynamic simulations. Explicit time integration is more accurate and efficient for simulations involving – Shock wave propagation – Large deformations and strains – Non-linear material behaviour – Complex contact – Fragmentation – Non-linear buckling.

Typical applications are drop tests, impact and penetration. ANSYS Explicit Dynamics analysis software provides simulation technology to help simulate structural performance long before manufacture.

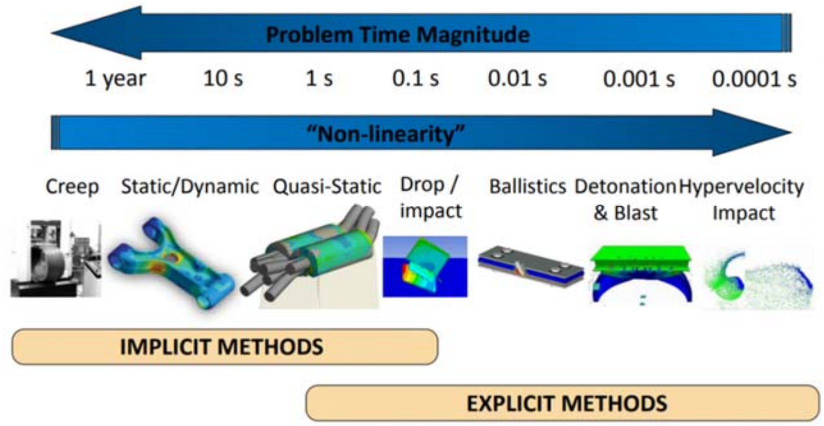
A time integration method used in Explicit Dynamics analysis system. It is so named because the method calculates the response at the current time using explicit information.

After defining the initial conditions (initial velocity, angular velocity), the analysis setting has to be maintained as per the problem requirement. In the analysis setting, time steps have to be defined explicitly, including:

- Initial time step
- Minimum time step
- Maximum time step
- Time step safety factor

In case of drop test the standard earth gravity is also taken into account.

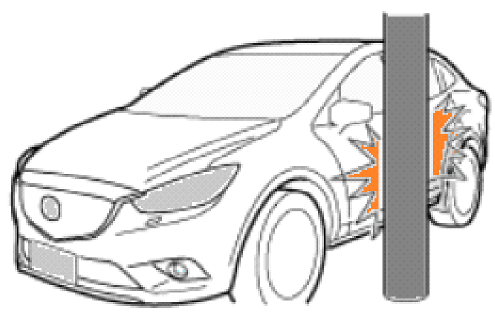
ANSYS Explicit Dynamics utilizes the Autodyn solver within the standard ANSYS Mechanical interface to analyses transient structural events and it is used for simulating fracture, cutting, failure, buckling, impact, drop as well as highly nonlinear quasi-static simulations that the implicit APDL-based solvers would struggle to converge.



https://www.mechead.com/what-is-explicit-dynamics-in-ansys/

A.2 Application description

In practice, there are many mechanical phenomena that are manifested by mechanical contacts and stresses made in very short periods of time, in the form of collisions. Some of these collisions can cause elastic deformation of the parts in contact, others can cause plastic deformation or even destruction and expulsion of material (in the case of penetration phenomena). In the field of motor vehicles, these dynamic impact requirements are very common. An analysis of the impact phenomena between a body element and a static element is very suggestive, and can be used in the design stages of body elements and passive safety elements.



A.3 Application goal

The application aims to determine the maximum values of the fields of displacements, deformations, internal stresses produced in collision on the component elements.

For this analysis, the use of two-dimensional elements was considered due to the simplicity of the geometric construction and the ease of modifying the profile of the studied elements.

B. PREPARATION OF THE MODEL FOR ANALYSIS

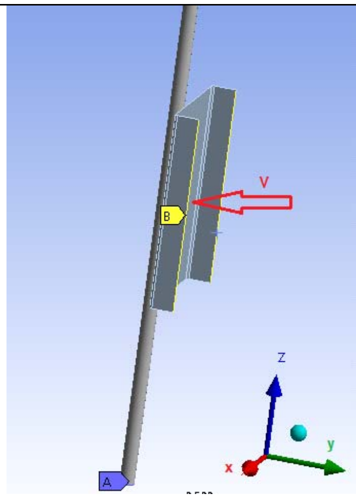
B.1 The model definition

In order to draw up the finite element analysis model associated with the present application, it is necessary to identify:

- geometric shape and dimensions,
- restrictions induced by links with adjacent elements,
- external and internal loads (own weight, speeds, accelerations),
- material characteristics.

B.2 The analysis model description

The analyzed model is intended to be very simple, consisting of only two elements: one of the tubular type which represents a fixed obstacle (steel pipe pillar with an average diameter of 60 mm and the wall of 5 mm) and the other, of the panel type by allumінium alloy 1 mm thick sheet, representing the moving object (dimensions are presented in Chapter C.3 - Geometric modeling). The collision with the vertical pole is considered to take place in the normal direction of the sheet metal panel, in the middle of it, according to the drawing below. The metal pillar is embedded at the bottom and the sheet metal panel moves in the direction of Ox in the direction of approaching the pillar with a speed of 25 m / s.



- point A – TX TY TZ RX RY RZ
- Vertical edges of the panel B – TX RY
- v_y = -25 m/s

B.3 Characteristics of the material

For finite element analysis the strength characteristics of the material, S235 steel are:

- modulus of longitudinal elasticity, E = 204,000 N / mm^2;
- transverse contraction coefficient (Poisson), ν = 0.3.

The characteristics of the second material, aluminum alloy, remain unchanged, according to the software library of materials.

- modulus of longitudinal elasticity, E = 75.000 N / mm^2;
- transverse contraction coefficient (Poisson), ν = 0.32.

C. PREPROCESSING OF FEA MODEL

C.1 Creating and saving the project

Create of the project

Toolbox: ↵ ⊟ Analysis Systems → ↵↵ Explicit Dynamics (the subproject window appears automatically); → [the name can be changed Explicit Dynamics în *Collision*].

Problem type setting (3D)

A: ↳ Geometry → ↵ Properties → Properties of Schematic A3: Geometry, ⊟ Advanced Geometry Options: ↵ Analysis Type, [select from list ↵, ↵ 3D] → [close window ↵ ✕].

Saving the project

↵ Save As... → Save As, File name: [input name, *Collision*] → ↵ Save.

C.2 Modelling of material characteristics

As there will be two separate parts, two different materials will be considered: steel and aluminum.

, Project Schematic: ↳ Engineering Data ✓ → ↵ Edit... → Outline of Schematic A2: Engineering Data:

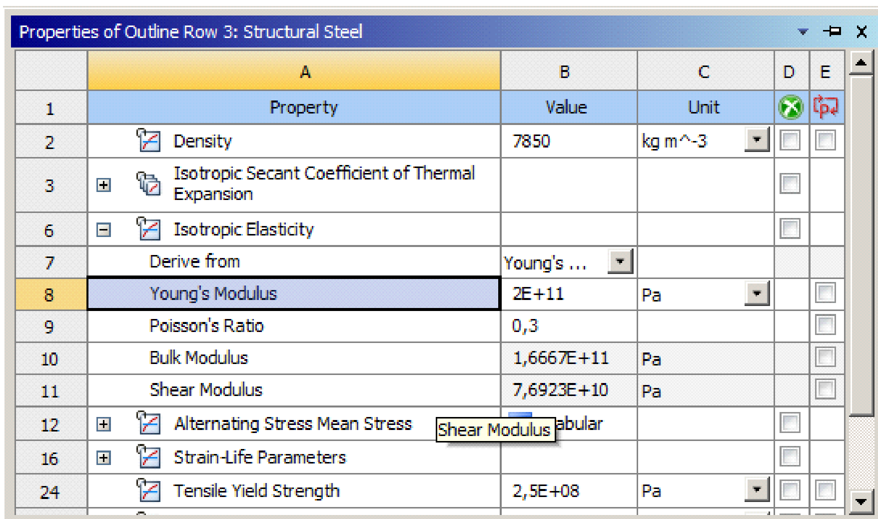
↵ Structural Steel , Properties of Outline Row 3: Structural Steel : ⊟ Isotropic Elasticity → Young's Modulus , Young's Modulus , [select from column C (Unit) cu ↵▼, ↵MPa], [input in column B (Unit) value, 204000] → ↵ Update Project → ↵ Return to Project (the other parameters remain the default).

To introduce the second material, follow the steps:

Engineering Data ✓ → Engineering Data Sources → Engineering Data Sources → General Materials → ☑ (se bifeaza căsuța / check box) →

Outline of ANSYS GRANTA Materials Data for Simulation (Sample) → Aluminum alloy, wrought, 6061, T6 (select Al alloy by check the box) → Outline of Schematic A2: Engineering Data (both materials are active: steel and aluminum).

If the window Properties of Outline Row 3: Structural Steel is not visible, the Outline and Properties or Reset Workspace options will be activated in the View menu, then the *Engineering Data Sources* command .



Properties of Outline Row 3: Structural Steel

	A	B	C	D	E
1	Property	Value	Unit		
2	Density	7850	kg m^-3		
3	⊞ Isotropic Secant Coefficient of Thermal Expansion				
6	⊟ Isotropic Elasticity				
7	Derive from	Young's ...			
8	Young's Modulus	2E+11	Pa		
9	Poisson's Ratio	0,3			
10	Bulk Modulus	1,6667E+11	Pa		
11	Shear Modulus	7,6923E+10	Pa		
12	⊞ Alternating Stress Mean Stress	Shear Modulus abular			
16	⊞ Strain-Life Parameters				
24	Tensile Yield Strength	2,5E+08	Pa		

C.3 Geometric modelling

C.3.1 Loading DesignModeler (DM) module

, Project Schematic: ↳ Geometry → ↵ New DesignModeler Geometry... → Units: ↵ ✓ Millimeter , ↵OK.

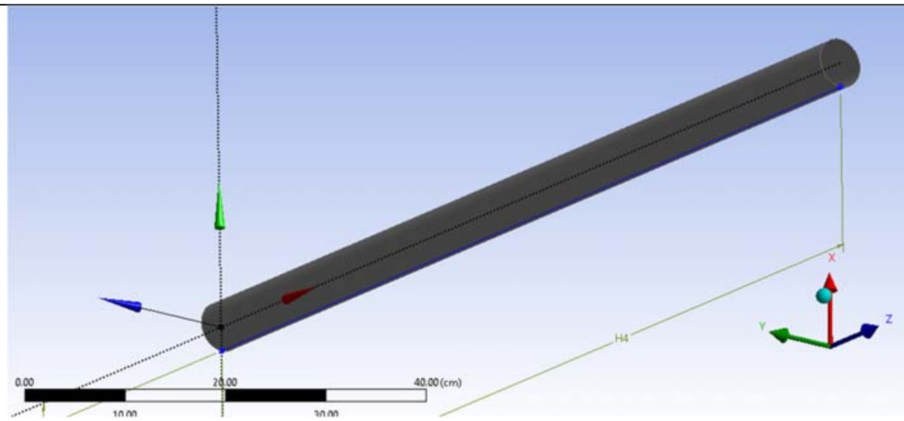
C.3.2 Pillar modelling

Construct a pipe with a circular section along the OZ axis, with surface-type elements, as follows:

Tree Outline → ✓ ZXPlane → Sketching (Look at plane) (draw a line parallel to the OZ axis, at distance 25 mm, 1000 mm long).

Dimensioning commands in the menu Dimensions will be used for sizing and positioning.

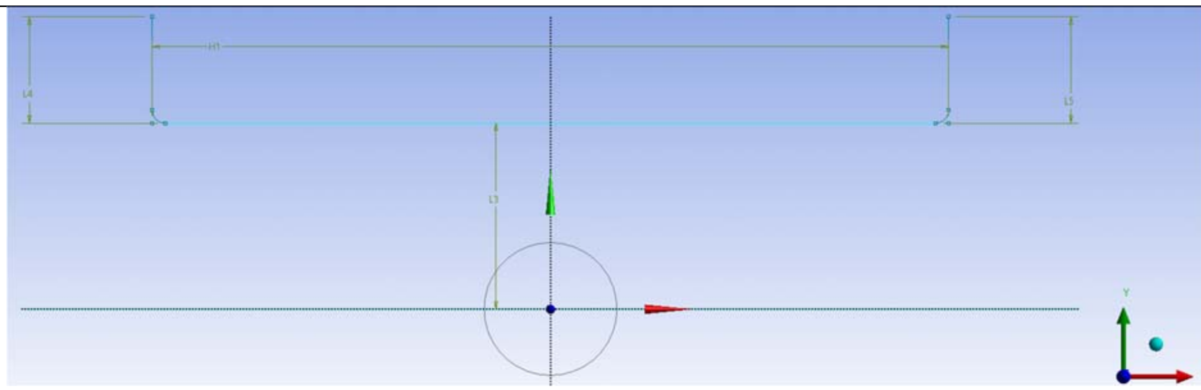
To create the 3D model, use the command Revolve , then rotate the drawn segment around the OZ axis, then Generate .



C.3.3 Panel modelling

A rectangular panel is constructed in a plane parallel to the XOZ, using surface-type elements. Since the panel will have reinforcements on two edges, its profile will be drawn in a plane perpendicular to the OZ and XOY axis, respectively.

Tree Outline → XYPlane → Sketching (Look at plane) Polyline (draw the profile in the figure below).



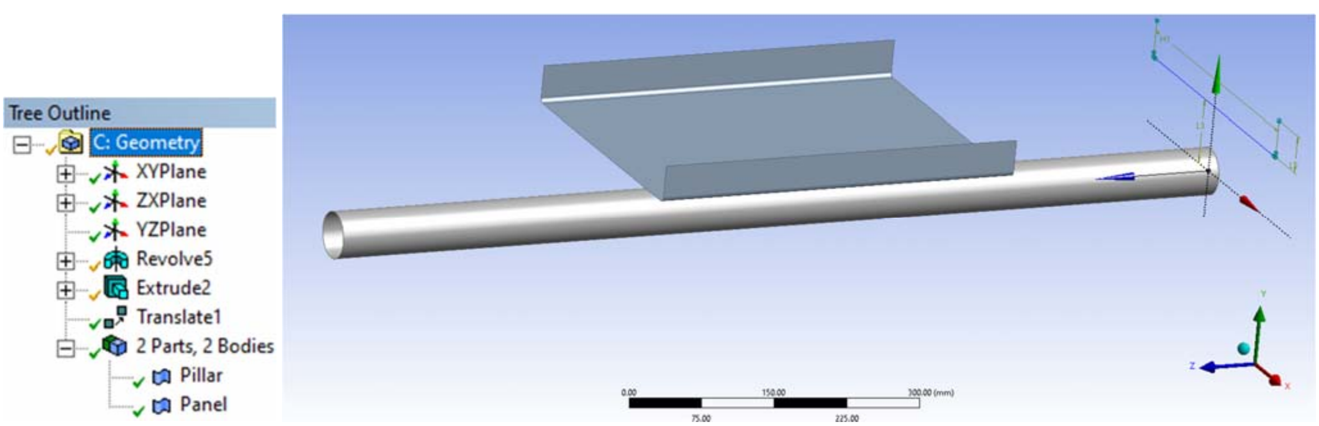
H1	300 mm
L3	70 mm
L4	40 mm
L5	40 mm

Modify

Fillet Radius: 5 mm

The profile will be extruded with the Extrude command in the OZ direction on the length of 400 mm, then it will be translated in the OZ direction at half the height of the pole, respectively 300 mm, using the commands:

Create → Body Transformation → Translate → Generate.



C.3.4 Saving the geometric model

→ (Save Project) → <u>File</u> → Close Design Modeler.

C.4 Finite element modelling

C.4.1 Activate the discretization module and set the finite element type

<u>Launching of the finite element modeling module</u>

, Project Schematic ↵↵ Model → launching module *Mechanical [ANSYS Multiphysics]*.

<u>Setting the unit of measurement system</u>

: ↵ Units → ↵ Metric (mm, kg, N, s, mV, mA) (the system of units of measurement is usually set by default).

<u>Setting the material characteristics</u>

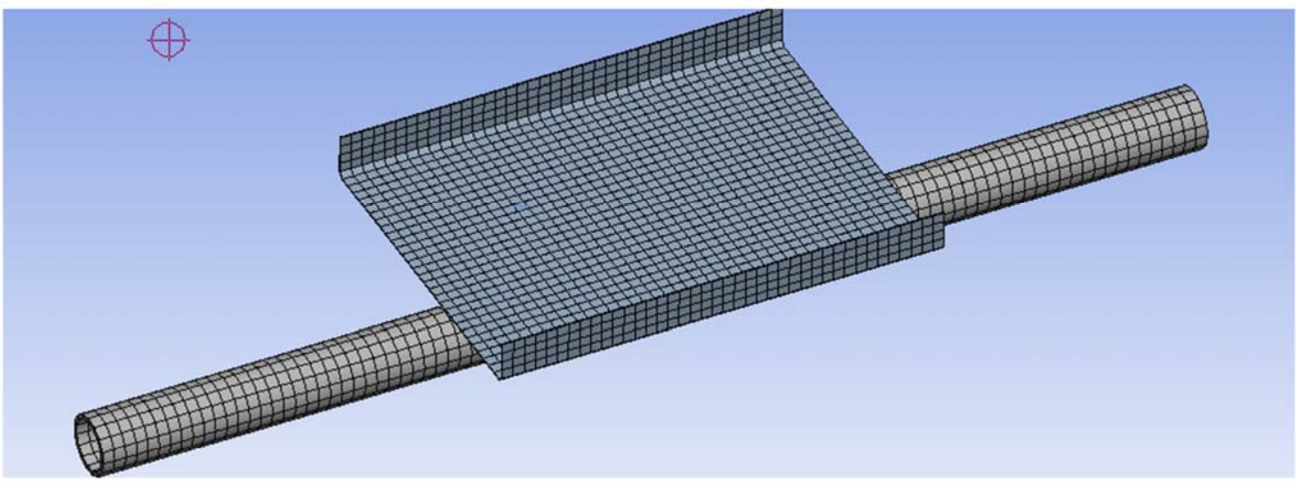
Outline: ↵ Geometry → Pillar → Details of "Pillar" → **Material** : ↵ Assignment → [select from list ↵, ↵ Structural Steel (default)] → **Definition** → Thickness 4.e-003 m.

Outline: ↵ Geometry → Panel → Details of "Panel" → **Material** : ↵ Assignment → [select from list ↵, ↵ Aluminum Alloy (default)] → **Definition** → Thickness 1.e-003 m.

C.4.2 Model discretization and finite element size setting

, Outline: ↵ Mesh → **Sizing** → Details of "Edge Sizing" - Sizing → **Scope** → Select Geometry: [will be selected with ↵ 3D geometry of the pillar, using the selection filter (Body)] Apply; **Definition Element** → Size: 10 mm → **Update** .

The operation will be repeated for the other body as well – Panel, Size: 10 mm.

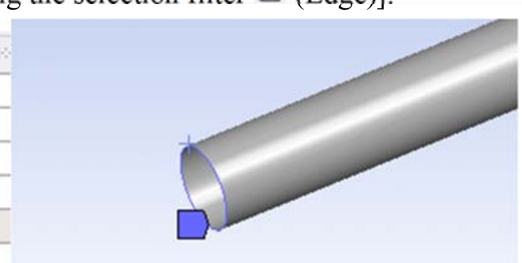


C.5 Modeling joints and constraints

<u>Inserting the fixed support</u>

, Outline: Explicit Dynamics ↵ Fixed → Details of "Fixed Support" → Geometry → Scope: [the circle at level 0 corresponding to the pillar is selected using the selection filter (Edge)].

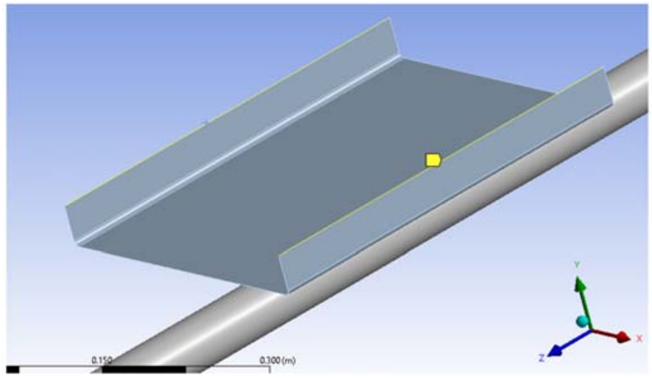
Details of "Fixed Support"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Type	Fixed Support
Suppressed	No



Enter the movement conditions of the panel

, Outline: ↲ Explicit Dynamics → Displacement Structural → Geometry → Scope: [select the edges parallel to the OZ axis of the panel using the selection filter (Edge)] → OX, OZ = 0, OY = free.

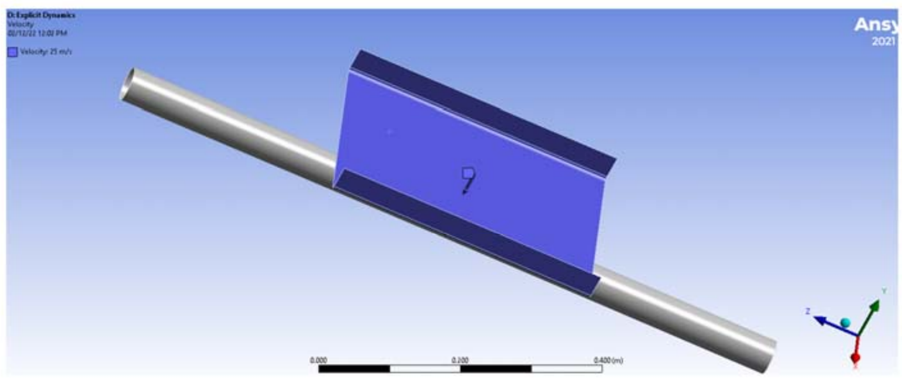
Details of "Displacement"	
Scope	
Scoping Method	Geometry Selection
Geometry	2 Edges
Definition	
Type	Displacement
Define By	Components
Coordinate System	Global Coordinate System
X Component	0. m (step applied)
Y Component	Free
Z Component	0. m (step applied)
Suppressed	No



C.6 Load modeling

Entering the speed of the panel

, Outline: Explicit Dynamics ↲ Initial Conditions → Velocity T=0 → Details of "Velocity" → **Scope** → Select Geometry: [will be selected with ↲ 3D geometry of the panel, using the selection filter (Body)] Apply; → **Definition** → Define by: Components → x = 0, y = -25 m/s, z = 0.



Introduction of analysis time

, Outline: Explicit Dynamics ↲ Analysis Settings → Details of "Analysis Settings" → **Step Controls** →

End Time: 0.01 s (estimate the travel time of the panel to the collision with the pole, depending on its initial distance and speed: $t = dist/speed = 70\ mm / 25\ m/s = 0.0028\ s$).

D. SOLVING THE FEA MODEL

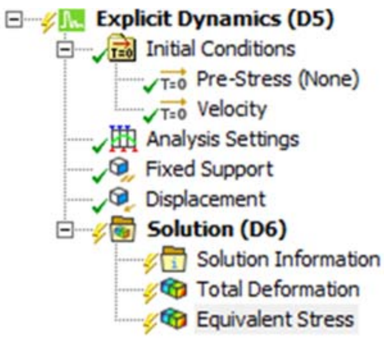
D.1 Launching the calculation module and select the types of results

In order to select the final data types to be analyzed after the launch of the calculation module, the series of commands presented below will be followed.

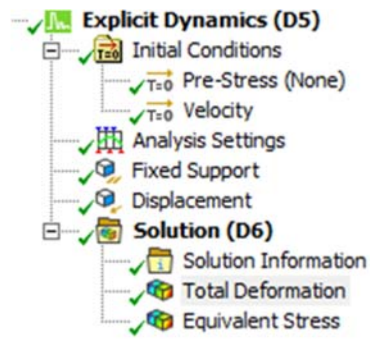
→ ↳ **Solution (D6)** → Insert → Deformation → Total [use the commands in the command box open with ↳].

The same result can be obtained by using commands:

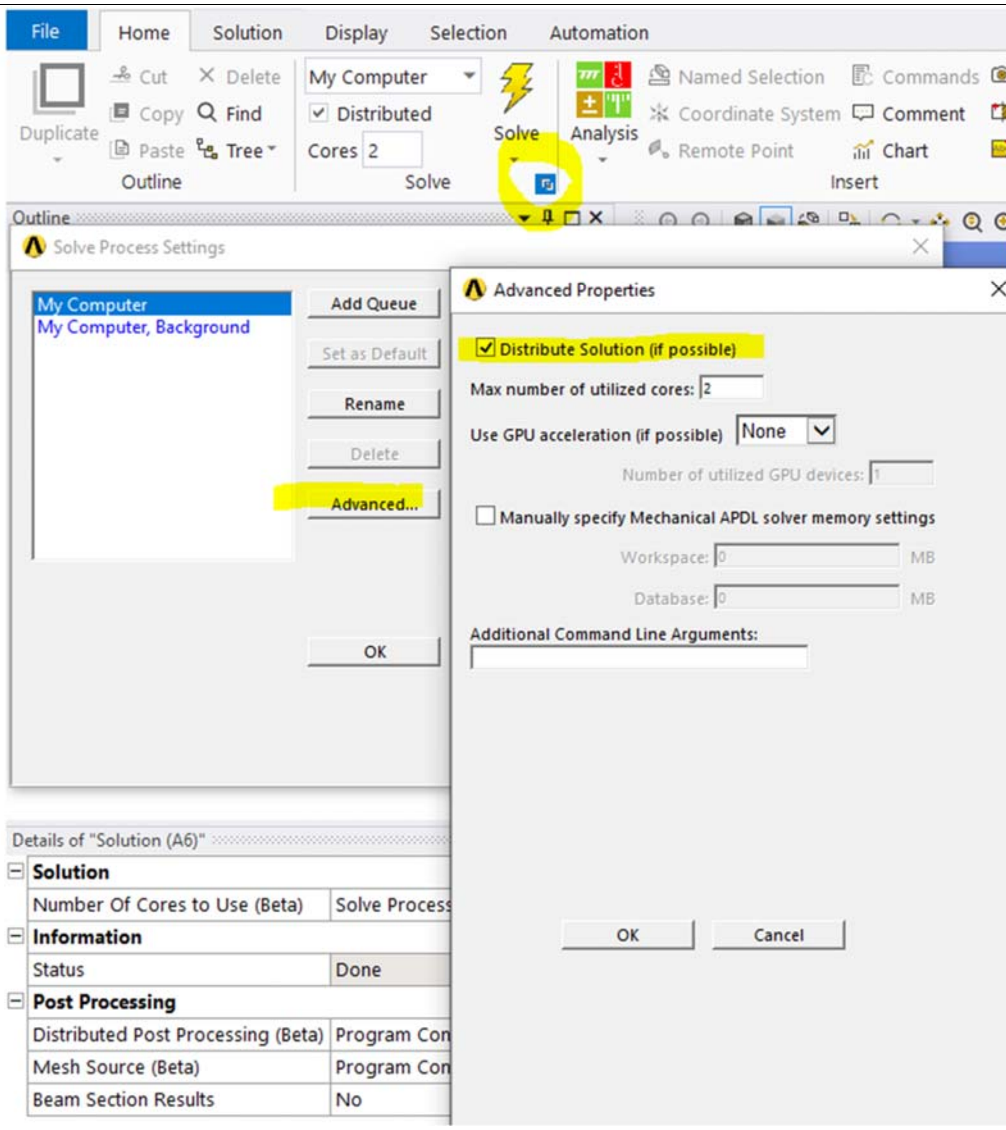
↲ Solution (D6) → **Solution** → Deformation → Total Deformation [the buttons in the menu bars are used] and ↲ Solution (D6) → **Solution** → Stress → Equivalent (von-Mises). ↲ Solution (A6) → Force Reaction → Tabular Data.
D.2. Launching the solving module
→ ↲ Solution (D6) → Solve.



→ Solve →



In case of errors in the processing of the model of the type "*An unknown error occurred during solution ...*", the solution below must be tried, by unchecking the highlighted button.



E. POST-PROCESSING OF RESULTS

E.1 Viewing the displacement fields

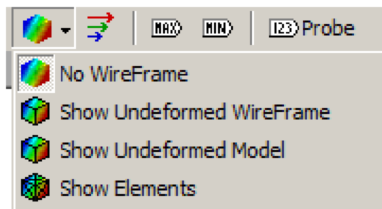
For suggestive results, set the view scale of the menu bars:
Result 8,6e+002 (Auto Scale) → Result 1.0 (True Scale).

Total deformation view

↲ Solution (A6) → Total Deformation → Graph → Animation.

If the images are not suggestive enough, in terms of how the work is distorted, you can return to changing the display scale by selecting a higher value: Result 1,7e+003 (2x Auto).

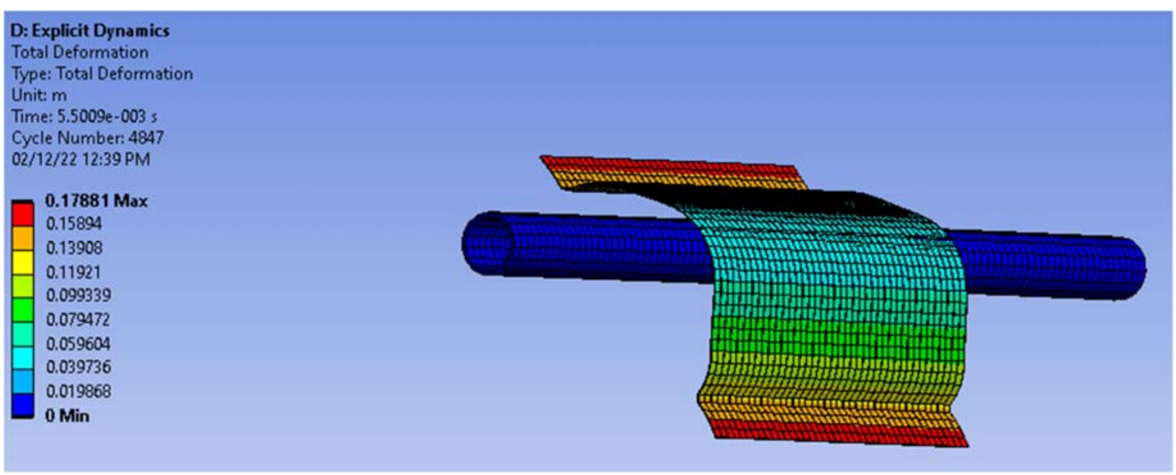
Various forms of distorted state representation can be used by calling the (Edge) button. Show Showformed WireFrame will be selected, an option that displays the undeformed and warped models in the same representation.



The display characteristics can be changed: the number of frames 10 Frames, as well as the running time of the simulation. At the same time, the result can be saved as a video file using the *Export Video File* command .

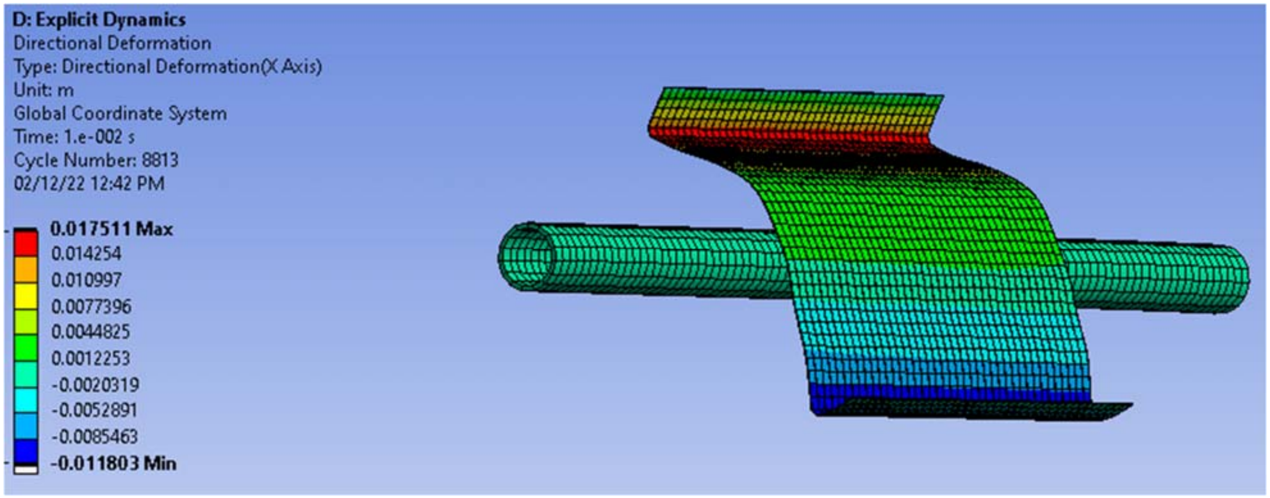
Total deformation view

↲ Solution (A6) → Total Deformation → Graph → Animation.



If you want to view it in another direction, follow the steps below:

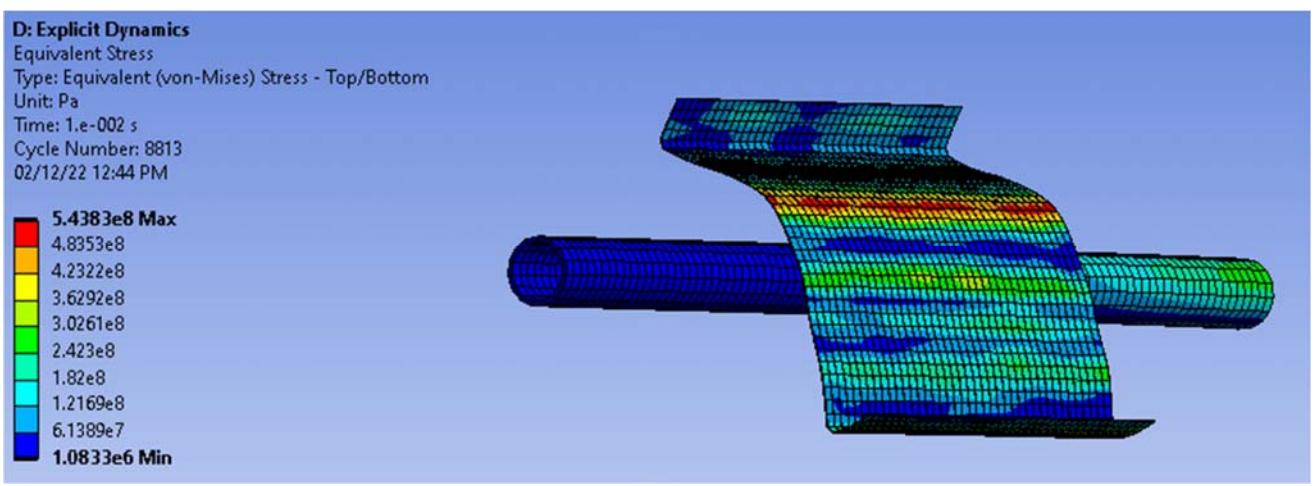
↲ Solution (A6) → Directional Deformation → Details of "Directional Deformation" → **Definition** → Orientation : Y Axis → Solve.

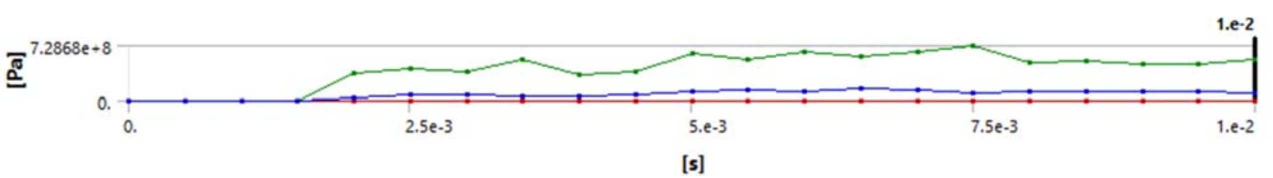


E.2. Visualizing the fields of stress, forces and moments

Equivalent Stress

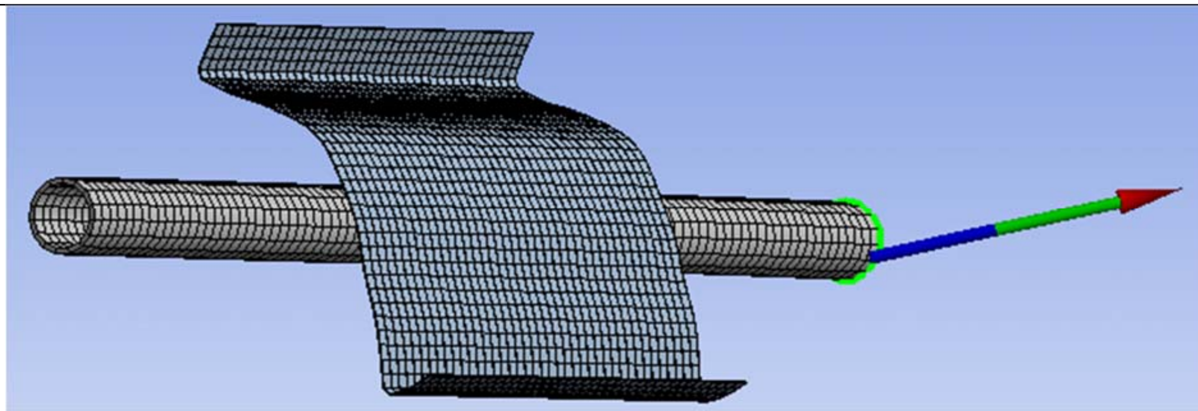
↲ Solution (A6) → Equivalent Stress → Graph → Animation.





Force reaction

↲ Solution (A6) → Force Reaction → Tabular Data.



Tabular Data

	Time [s]	Force Reaction (X) [N]	Force Reaction (Y) [N]	Force Reaction (Z) [N]	Force Reaction (Total)
13	6.0003e-003	-328.63	2307.3	659.87	2422.2
14	6.5007e-003	464.37	5059.5	-4897.5	7056.9
15	7.001e-003	335.72	6727.6	-1172.2	6837.2
16	7.5002e-003	-461.69	2355.4	-606.84	2475.8
17	8.0005e-003	417.92	483.06	2291.	2378.3
18	8.5008e-003	-43.553	1236.6	2040.5	2386.4
19	9.0009e-003	-803.21	-326.26	-1053.	1364.
20	9.5011e-003	139.2	2150.4	1931.9	2894.1

F. RESULTS ANALYSIS

It is observed that, despite the fact that the modeling of the parts was performed with the help of surface type bodies, the results obtained are suggestive, being presented in a 3D environment, due to the ease of the program used to attach various thicknesses to the structure.

Changing the thickness and materials of the various components is very easy to do, this can be done even at the end of an analysis, and after an update order, the results of the new analysis will change according to the new initial conditions.

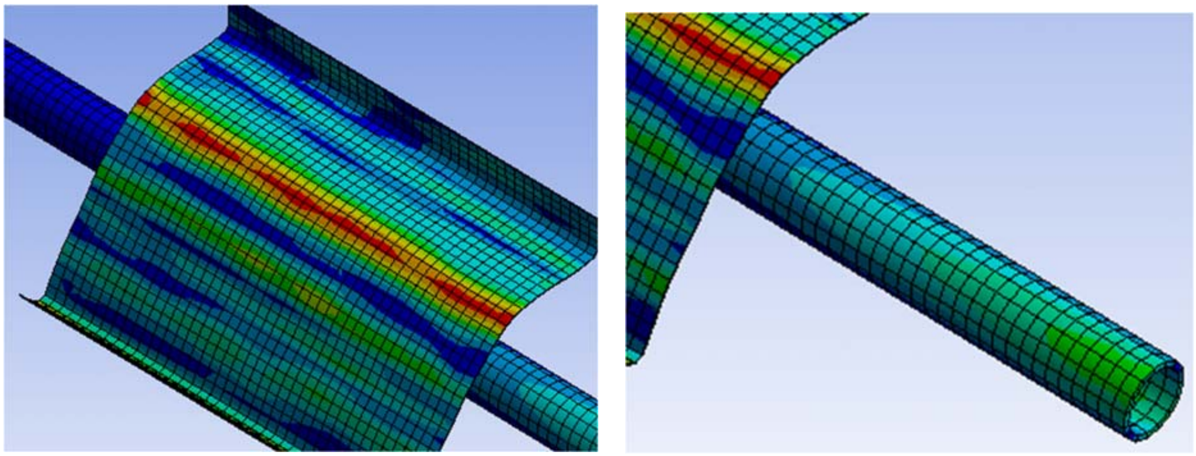
From the point of view of the total deformations, it is observed that the maximum value is 0.0175 m in the area of the vertical edges, the entities furthest from the center of the panel.

Examining the graphical representation of the total equivalent stresses (fig.a), it is observed that the panel is strongly stressed in the areas of contact with the column, reaching values (543 MPa) that exceed the elastic limit of the material, entering the flow area (σ_c=280 Mpa).

In other words, at these initial data (constraints and speed of the panel) a plastic deformation of the panel can be noticed.

From the point of view of the tubular pillar, made of steel (fig. b), a voltage of values of 180-200 Mpa is observed, below the flow limit. So, the pillar will deform elastically, returning to its original shape and dimensions.

If the initial speed changes from 25 to 10 m / s, it will be noticed that the value of the stresses generated by the impact is lower (280 Mpa), as well as the deformation of the panel (0.005 m), remaining in the area of elastic deformations.



a. *b.*

G. CONCLUSIONS

From the point of view of the pre-processing phase, it can be seen that the use of 2D bodies involves minimal resources for both modeling and discretization. Another strong point is that the thickness of the parts (either of the column or of the sheet metal panel) can be modified very easily, without influencing the basic shape of the bar structure.

The introduction of supports, constraints and demands is quick and easy. The declaration of materials, as well as the discretization of the structure in the form of surfaces are controllable processes, and can be done automatically or manually.

Analyzing the results obtained by FEM, it can be seen that it provides much more data, at a time and with much lower resource consumption, than the analytical version.