

# S01t.- Introducción curso ANSYS.

Mejora 2122....

# • Presentación profesor



## Dr. Andrés-Amador García Granada

- Ingeniero industrial por ETSEIB UPC especialidad Mecánica de Estructuras (1996) y PhD University of Bristol (2000)
- Ha trabajado siempre en temas de simulación en EDAG, SEAT y AUDI entre otras en las que realizaba las simulaciones y ensayos de choques con la creación de la base de materiales para poder llevarlos a cabo.
- Desde el 2007 trabaja como profesor en IQS siendo jefe de departamento de ingeniería industrial desde octubre de 2018.



## • Presentación curso

- Necesidades de espacio para frenar un impacto. Diferencia entre impacto elástico y plástico. Discusión sobre la rotura. Comportamiento del material a compresión o flexión. Vibraciones, Pandeo, Influencia de temperatura, Creep.
- Obtención de curvas de material para estimar espacios y hacer simulaciones. Tratamiento de ensayo a tracción y de flexión. Ensayo charpy.
- Modelos de material disponibles en ANSYS Diferencias entre cálculos explícitos e implícitos. Unidades coherentes. Importancia de la malla y del time step. Mallado SHELL. Posibilidades de pre-procesador industrial como ANSA.
- Cálculo de costes con SolidWorks Costing y Topology optimisation.
- Cálculos de impacto, vibraciones, rigidez, creep.
- Resumen de curso introductorio para centrar las necesidades en un curso práctico avanzado si se considerara necesario.

## • Presentación curso

- La formación, con un total de 10h, será 100% online a repartir en 5 sesiones de 2horas cada una.

- Test inicial

$$5[\text{m/s}] \cdot 3600[\text{s/h}] / 1000[\text{m/km}] = 18[\text{km/h}] \quad 9\text{de}9$$

$$m \cdot v^2 / 2 = 20[\text{kg}] \cdot 5^2 / 2[\text{m}^2/\text{s}^2] = 250[\text{N.m}] = 250[\text{J}] \quad 9\text{de}9$$

## Test comparativo crash

\*Obligatorio

Nombre y Apellidos \*

Tu respuesta

Email de contacto \*

Tu respuesta

¿Cuántos [km/h] equivalen a 5[m/s]?

Tu respuesta

¿Qué energía cinética tiene un paquete de 20[kg] que cae a 5[m/s]?

Tu respuesta

$$v = \sqrt{2gh} = \sqrt{2 \cdot 10 \cdot 1.25} = 5 \text{ [m/s]} \quad 8 \text{ de } 9$$

$$t = v/a = 5/100 = 0.05 \text{ [s]} \quad 8 \text{ de } 9$$

$$s = at^2/2 = v^2/(2a) = 5^2/(2 \cdot 100) = 0.125 \text{ [m]} \quad 4 \text{ de } 9$$

Estiramos  $0.01/100 = 0.0001$  y por tanto  
 $\sigma = E\varepsilon = 200000 \text{ [MPa]} \cdot 0.0001 = 20 \text{ [MPa]}$  no  
 plastifica

$$F = \sigma A = 20 \cdot 10 \cdot 1 = 200 \text{ [N]} \quad 3 \text{ de } 9$$

Estiramos  $1/100 = 0.01$  y por tanto  
 $\sigma = E\varepsilon = 200000 \text{ [MPa]} \cdot 0.01 = 2000 \text{ [MPa]}$  que  
 plastifica y por tanto plastifica  $\sigma = 200 \text{ [MPa]}$   
 $F = \sigma A = 200 \cdot 10 \cdot 1 = 2000 \text{ [N]} \quad 0 \text{ de } 9$

¿Qué velocidad impacta un paquete que cae desde 1.25[m] de altura sin rozamientos considerando  $g = 10 \text{ [m/s}^2\text{]}$ ?

Tu respuesta

¿Cuánto tiempo tarda un paquete en pasar de 5m/s a 0 si se le aplica una aceleración constante de  $100 \text{ [m/s}^2\text{]}$ ?

Tu respuesta

¿Cuánto espacio necesitamos para parar un paquete y pasar de 5m/s a 0 si se le aplica una aceleración constante de  $100 \text{ [m/s}^2\text{]}$ ?

Tu respuesta

Si tenemos una pieza de acero que plastifica a  $2000 \text{ MPa}$  de forma perfecta con  $L_x = 100 \times L_y = 10 \times L_z = 1 \text{ [mm]}$  y estiramos en  $x$   $0.01 \text{ mm}$  ¿Qué fuerza aplicamos?

Tu respuesta

Si tenemos una pieza de acero que plastifica a  $200 \text{ MPa}$  de forma perfecta con  $L_x = 100 \times L_y = 10 \times L_z = 1 \text{ [mm]}$  y estiramos en  $x$   $1 \text{ mm}$  ¿Qué fuerza aplicamos?

$$E=kx^2/2=1000000*(0.001)^2/2=0.5[J] \text{ 6de9}$$

Encontrar propiedades de molded pulp es difícil, En <sup>(1)</sup>el módulo de Young es aproximadamente 1200[MPa] mientras <sup>(2)</sup>da la curva entera plastificando a unos 20[MPa] **0de9**.

$$0.1/0.0001=1000 \text{ pasos de cálculo } \text{2de9}$$

Para hacer algo de un espesor de 1[mm] como sólido el tamaño de malla es muy pequeño y el time step pequeño y número de elementos Elevado impide que acabe el cálculo. **1de9**

En un muelle de  $k=1000000[N/m]$  ¿Qué energía almacena al comprimir 1[mm]

Tu respuesta

---

Dar el módulo de Young de molded pulp

Tu respuesta

---

Para calcular 0.1[s] con un time step de 0.1[us] ¿Cuántos pasos de cálculo necesito?

Tu respuesta

---

¿Por que usan modelo 2D tipo SHELL en simulaciones de automoción, trenes, aeronautica?

Tu respuesta

---

<sup>(1)</sup>[https://www.researchgate.net/publication/347339887\\_The\\_impact\\_of\\_molded\\_pulp\\_product\\_process\\_on\\_the\\_mechanical\\_properties\\_of\\_molded\\_BCTMP/fulltext/5fe13c44299bf140883332cf/The-impact-of-molded-pulp-product-process-on-the-mechanical-properties-of-molded-BCTMP.pdf](https://www.researchgate.net/publication/347339887_The_impact_of_molded_pulp_product_process_on_the_mechanical_properties_of_molded_BCTMP/fulltext/5fe13c44299bf140883332cf/The-impact-of-molded-pulp-product-process-on-the-mechanical-properties-of-molded-BCTMP.pdf)

<sup>(2)</sup>[https://www.researchgate.net/publication/339505304\\_Poisson%27s\\_ratios\\_of\\_molded\\_pulp\\_materials\\_by\\_digital\\_image\\_correlation\\_method\\_and\\_uniaxial\\_tensile\\_test](https://www.researchgate.net/publication/339505304_Poisson%27s_ratios_of_molded_pulp_materials_by_digital_image_correlation_method_and_uniaxial_tensile_test)

Dar el time step estable para una malla de 1[mm] de acero

Según <sup>(3)</sup>el incremento se calcula: **0de9**

$$\Delta t \leq \min \left( L_e \sqrt{\frac{\rho}{\hat{\lambda} + 2\hat{\mu}}} \right) \quad \Delta t = kL \sqrt{\delta / E}$$

k	[-]	1	1	1	0.9
		Steel	Aluminium	PP-LGF	Steel
E	[GPa]	210	70	3	210
d	[kg/mm <sup>3</sup> ]	7.85E-06	2.80E-06	1.04E-06	7.85E-06
Time step for L10	[μs]	1.93E+00	2.00E+00	5.89E+00	1.74E+00
Time step for L5	[μs]	9.67E-01	1.00E+00	2.94E+00	8.70E-01
L for time 1μs	[mm]	5.17E+00	5.00E+00	1.70E+00	5.75E+00
L_for_time_0.5μs	[mm]	2.59E+00	2.50E+00	8.49E-01	2.87E+00

<sup>(1)</sup>[https://www.researchgate.net/publication/347339887/The\\_impact\\_of\\_molded\\_pulp\\_product\\_process\\_on\\_the\\_mechanical\\_properties\\_of\\_molded\\_BCTMP/fulltext/5fe13c44299bf140883332cf/The-impact-of-molded-pulp-product-process-on-the-mechanical-properties-of-molded-BCTMP.pdf](https://www.researchgate.net/publication/347339887/The_impact_of_molded_pulp_product_process_on_the_mechanical_properties_of_molded_BCTMP/fulltext/5fe13c44299bf140883332cf/The-impact-of-molded-pulp-product-process-on-the-mechanical-properties-of-molded-BCTMP.pdf)

<sup>(2)</sup>[https://www.researchgate.net/publication/339505304/Poisson%27s\\_ratios\\_of\\_molded\\_pulp\\_materials\\_by\\_digital\\_image\\_correlation\\_method\\_and\\_uniaxial\\_tensile\\_test](https://www.researchgate.net/publication/339505304/Poisson%27s_ratios_of_molded_pulp_materials_by_digital_image_correlation_method_and_uniaxial_tensile_test)

<sup>(3)</sup> <https://abaqus-docs.mit.edu/2017/English/SIMACAEANLRefMap/simaanl-c-expdynamic.htm>



## • Ejemplo inicial

Vamos a estirar 1mm una barra de 1x20x300[mm] de acero en 1[ms].  
 (A=20x1=20[mm<sup>2</sup>], L=300[mm])

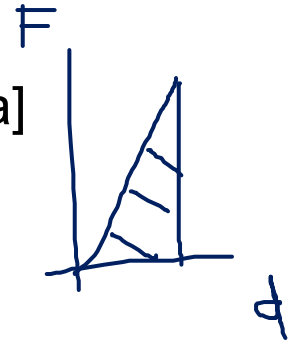
**Estática:**  $F = EADL/L$

deformación:  $\varepsilon = \Delta L/L = 1/300 = 0.003333[-] = 0.33\%$

tensión:  $\sigma = E\varepsilon = 200[\text{GPa}] * 0.003333 = 0.6667[\text{GPa}] = 666.7[\text{MPa}]$

fuerza:  $F = \sigma A = 666.7 * 20 * 1 = 13333[\text{N}] = 13.33[\text{kN}]$

energía:  $E = F\Delta L/2 = 13333 * 0.001/2 = 6.667[\text{J}] = 6.667[\text{kN}\cdot\text{mm}]$



**Dinámica:**  $F = EA\Delta L/L + ma$

aceleración:  $a = 2 * \Delta L / (t^2) / 2 = 1[\text{mm}/\text{ms}^2] = 1000[\text{m}/\text{s}^2]$

masa:  $m = 7.85e-6[\text{kg}/\text{mm}^3] * 300 * 20 * 1[\text{mm}^3] = 4.71e-2[\text{kg}]$

fuerza inercial:  $F_i = ma = 47.1[\text{N}]$

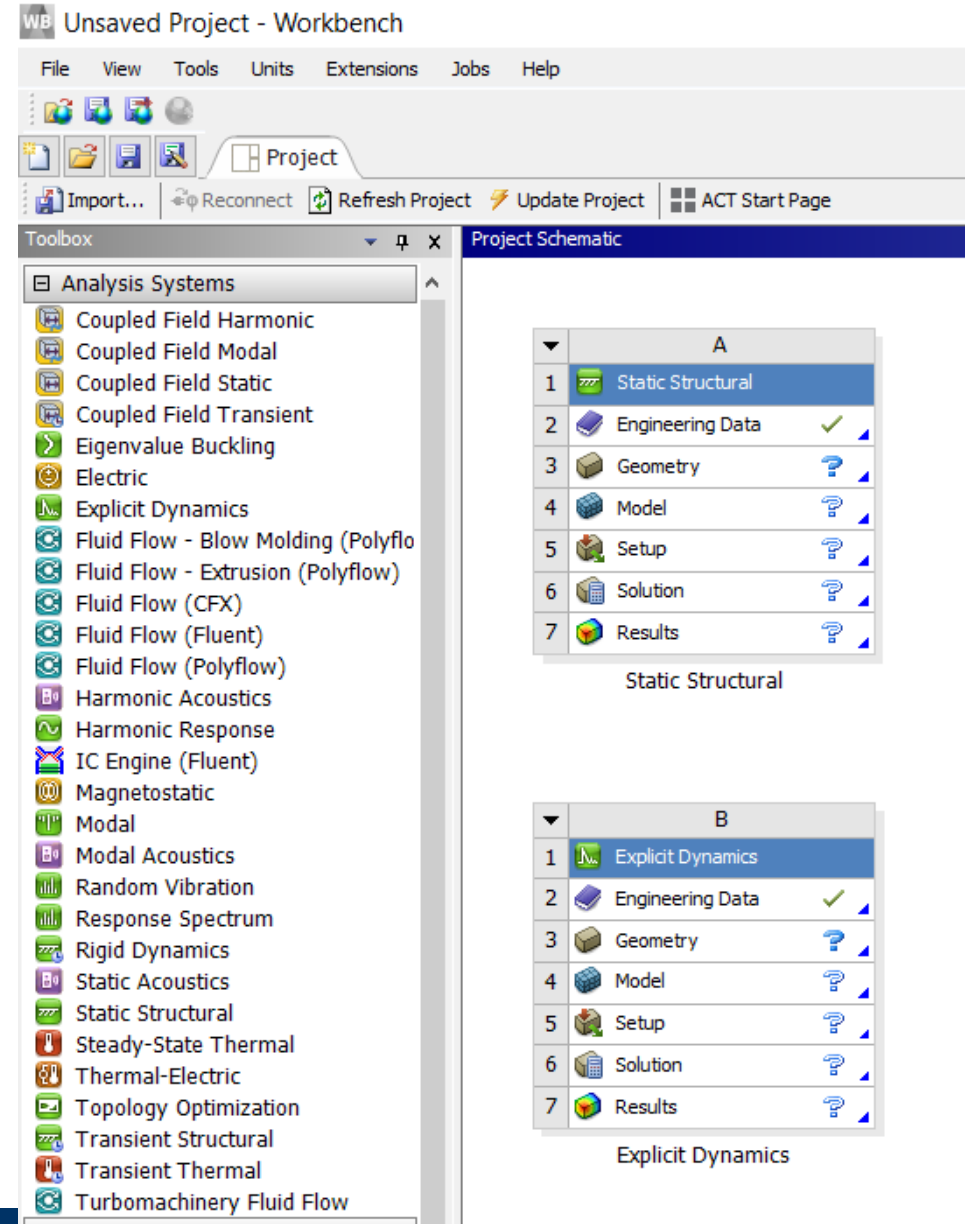
velocidad:  $v = a * t = 1[\text{m}/\text{s}] = 1[\text{mm}/\text{ms}]$

energía cin.:  $0.5 * m * v^2 = 0.02352[\text{J}] = 0.002352[\text{kN}\cdot\text{mm}]$

Para estas masas y aceleraciones la fuerza inercial es muy pequeña en comparación a la fuerza de deformación.

# • Presentación Asignatura

Se pretende hacer un pequeño curso de uso de ANSYS Workbench comparando estática y dinámica



# • Propiedades de material en Engineering data

01-inicio - Workbench

File Edit View Tools Units Extensions Jobs Help

Project A2:Engineering Data

Filter Engineering Data Engineering Data Sources

Toolbox

Field Variables

Outline of Schematic A2: Engineering Data

	A	B	C	D	E
1	Contents of Engineering Data	Source			Description
2	Material				
3	Structural Steel				Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5 -110.1
*	Click here to add a new material				

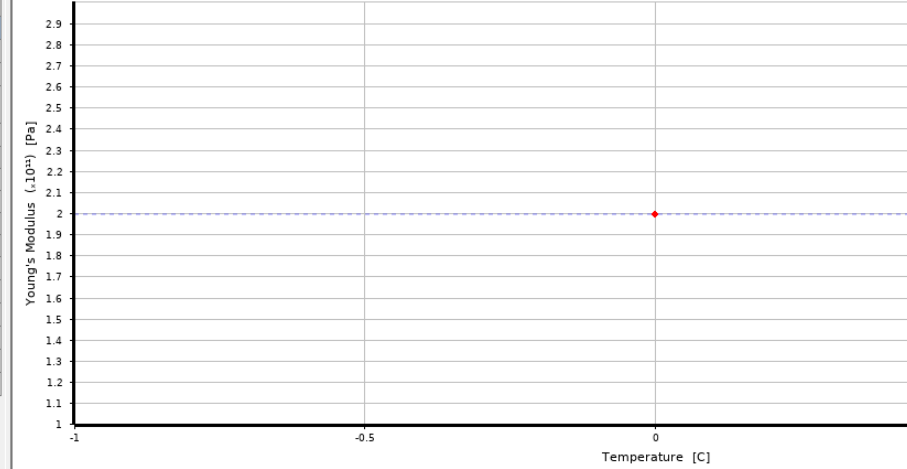
Table of Properties Row 6: Isotropic Elasticity

	A	B	C	D	E
1	Temperature (C)	Young's Modulus (Pa)	Poisson's Ratio	Bulk Modulus (Pa)	Shear Modulus (Pa)
2		2E+11	0.3	1.6667E+11	7.6923E+10
*					

Properties of Outline Row 3: Structural Steel

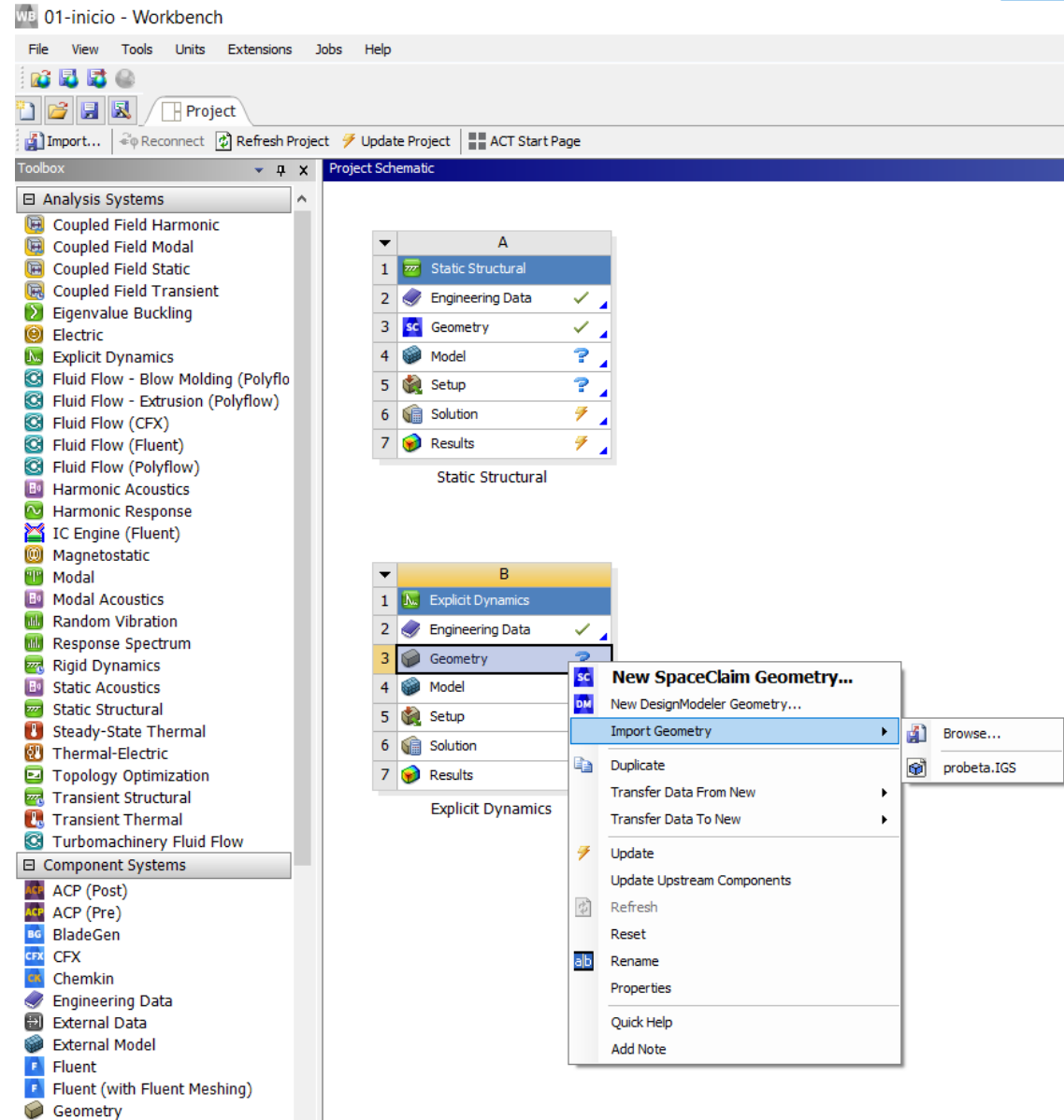
	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Density	7850	kg m <sup>-3</sup>		
4	Isotropic Secant Coefficient of Thermal Expansion				
6	Isotropic Elasticity				
7	Derive from	Young's ...			
8	Young's Modulus	2E+11	Pa		
9	Poisson's Ratio	0.3			
10	Bulk Modulus	1.6667E+11	Pa		
11	Shear Modulus	7.6923E+10	Pa		
12	Strain-Life Parameters				
20	S-N Curve	Tabular			
24	Tensile Yield Strength	2.5E+08	Pa		
25	Compressive Yield Strength	2.5E+08	Pa		
26	Tensile Ultimate Strength	4.6E+08	Pa		
27	Compressive Ultimate Strength	0	Pa		

Chart of Properties Row 6: Isotropic Elasticity



# • Geometría

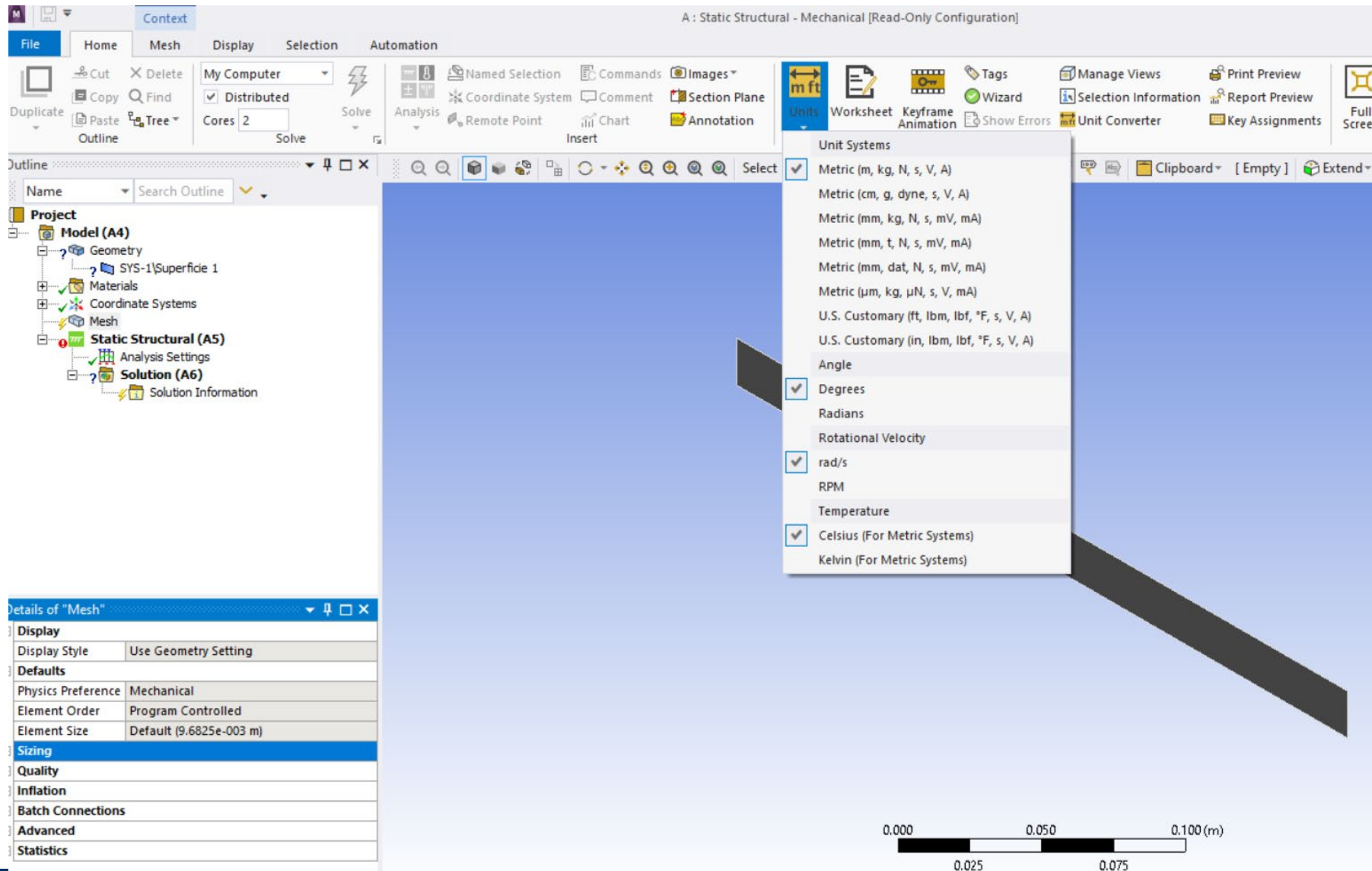
Importar o dibujar en SpaceClaim



The screenshot displays the ANSYS Workbench interface for a project named '01-inicio - Workbench'. The 'Toolbox' on the left lists various analysis systems, including 'Analysis Systems' (e.g., Static Structural, Explicit Dynamics) and 'Component Systems' (e.g., ACP, BladeGen, Chemkin). The 'Project Schematic' on the right shows two analysis systems, 'A' (Static Structural) and 'B' (Explicit Dynamics). In system 'B', the 'Geometry' component is selected, and a context menu is open, showing options like 'New SpaceClaim Geometry...', 'Import Geometry', 'Duplicate', and 'Update'. The 'Import Geometry' option is highlighted, and a 'Browse...' dialog box is visible next to it, indicating the process of importing a geometry file (e.g., 'probeta.IGS').

# • Modelo

Editar



The screenshot displays the ANSYS Workbench interface for a Static Structural analysis. The main window shows a 3D model of a mechanical part with a mesh. The 'Units' menu is open, showing various unit systems. The 'Metric (m, kg, N, s, V, A)' system is selected. Other options include 'Metric (cm, g, dyne, s, V, A)', 'Metric (mm, kg, N, s, mV, mA)', 'Metric (mm, t, N, s, mV, mA)', 'Metric (mm, daN, N, s, mV, mA)', 'Metric (μm, kg, μN, s, V, A)', 'U.S. Customary (ft, lbf, lbf, °F, s, V, A)', 'U.S. Customary (in, lbf, lbf, °F, s, V, A)', 'Angle', 'Degrees', 'Radians', 'Rotational Velocity', 'rad/s', 'RPM', 'Temperature', 'Celsius (For Metric Systems)', and 'Kelvin (For Metric Systems)'. The 'Details of Mesh' panel is visible in the bottom left, showing settings for Display, Defaults, Sizing, Quality, Inflation, Batch Connections, Advanced, and Statistics. A scale bar at the bottom right indicates dimensions from 0.000 to 0.100 (m).

Unit Systems

- Metric (m, kg, N, s, V, A)
- Metric (cm, g, dyne, s, V, A)
- Metric (mm, kg, N, s, mV, mA)
- Metric (mm, t, N, s, mV, mA)
- Metric (mm, daN, N, s, mV, mA)
- Metric (μm, kg, μN, s, V, A)
- U.S. Customary (ft, lbf, lbf, °F, s, V, A)
- U.S. Customary (in, lbf, lbf, °F, s, V, A)
- Angle
- Degrees
- Radians
- Rotational Velocity
- rad/s
- RPM
- Temperature
- Celsius (For Metric Systems)
- Kelvin (For Metric Systems)

Details of "Mesh"

Display	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	Mechanical
Element Order	Program Controlled
Element Size	Default (9.6825e-003 m)
Sizing	
Quality	
Inflation	
Batch Connections	
Advanced	
Statistics	

0.000 0.025 0.050 0.075 0.100 (m)

## • Unidades

En grupo VW se utilizan las unidades marcadas en este cuadro para Implícito (Abaqus y Nastran) y explícito (PamCrash)

basic units			derived units		
LENGTH	MASS	TIME	FORCE	STRESS	ENERGY
cm	g	s	dyn=g cm/s <sup>2</sup>	dyn/cm <sup>2</sup>	dyn cm=erg
m	kg	s	N=kg m/s <sup>2</sup>	N/m <sup>2</sup> =Pa	N m=J
cm	100 kg	s	N=100 kg cm/s <sup>2</sup>	N/cm <sup>2</sup> =10 <sup>-1</sup> bar	cm
mm	ton	s	N=ton mm/s <sup>2</sup>	N/mm <sup>2</sup> =MPa	N mm ← implícito
mm	kg	ms	kN=kg mm/ms <sup>2</sup>	kN/mm <sup>2</sup> =GPa	kN mm ← explícito
mm	dton	s	kgf=9.81 ton mm/s <sup>2</sup>	kgf/mm <sup>2</sup>	kgf mm
cm	g	μs	10 <sup>7</sup> N=g cm/μs <sup>2</sup>	10 <sup>7</sup> N/cm <sup>2</sup> =Mbar	10 <sup>7</sup> N cm
ft	slug	s	lbf (or pf) = slug ft/s <sup>2</sup>	lbf/ft <sup>2</sup> (or pf/ft <sup>2</sup> )	lbf ft (or pf ft),
mm	kg/G	ms	10 <sup>3</sup> kgf=metric ton-f	9.81 GPa	9.81 kN mm

Se ha de ir con mucho cuidado al pasar los modelos de un software a otro, incluso con las cartas de materiales.

# • Modelo

Editar

A : Static Structural - Mechanical [ANSYS Mechanical Enterprise]

File Home Geometry Display Selection Automation

Duplicate Outline Solve Analysis Remote Point Coordinate System Comment Chart Annotation Insert

Replace Geometry Point Distributed Transform Part Surface Coating Element Orientation Virtual Body Virtual

Outline Name Search Outline

Project\*

- Model (A4)
  - Geometry
    - SYS-1\Superficie 1
  - Materials
  - Coordinate Systems
  - Mesh
  - Static Structural (A5)
    - Analysis Settings
    - Solution (A6)
      - Solution Information

etails of "SYS-1\Superficie 1"

Graphics Properties	
<b>Definition</b>	
<input type="checkbox"/> Suppressed	No
Dimension	3D
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
<input checked="" type="checkbox"/> Thickness	1.e-003 m
Thickness Mode	Manual
Offset Type	Middle
Treatment	None
Model Type	Shell
<b>Material</b>	



# • Modelo

## Editar

Context A : Static Structural - Mechanical [Read-Only Configuration]

File Home Materials Display Selection Automation

Duplicate Outline Solve Solvers Analysis Remote Point Coordinate System Comment Chart Insert Images Section Plane Annotation Material Assignment Material Plot Material Combination

Outline Engineering Data: Material View

**Structural Steel**

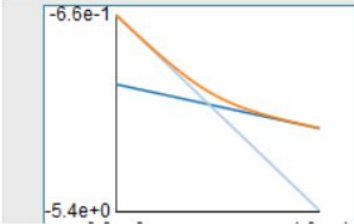
Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1

Density	7850 kg/m <sup>3</sup>
---------	------------------------

**Structural**

<b>Isotropic Elasticity</b>	
Derive from	Young's Modulus and Poisson's Ratio
Young's Modulus	2e+11 Pa
Poisson's Ratio	0.3
Bulk Modulus	1.6667e+11 Pa
Shear Modulus	7.6923e+10 Pa
Isotropic Secant Coefficient of Thermal Expansion	1.2e-05 1/°C
Compressive Ultimate Strength	0 Pa
Compressive Yield Strength	2.5e+08 Pa

Strain-Life Parameters



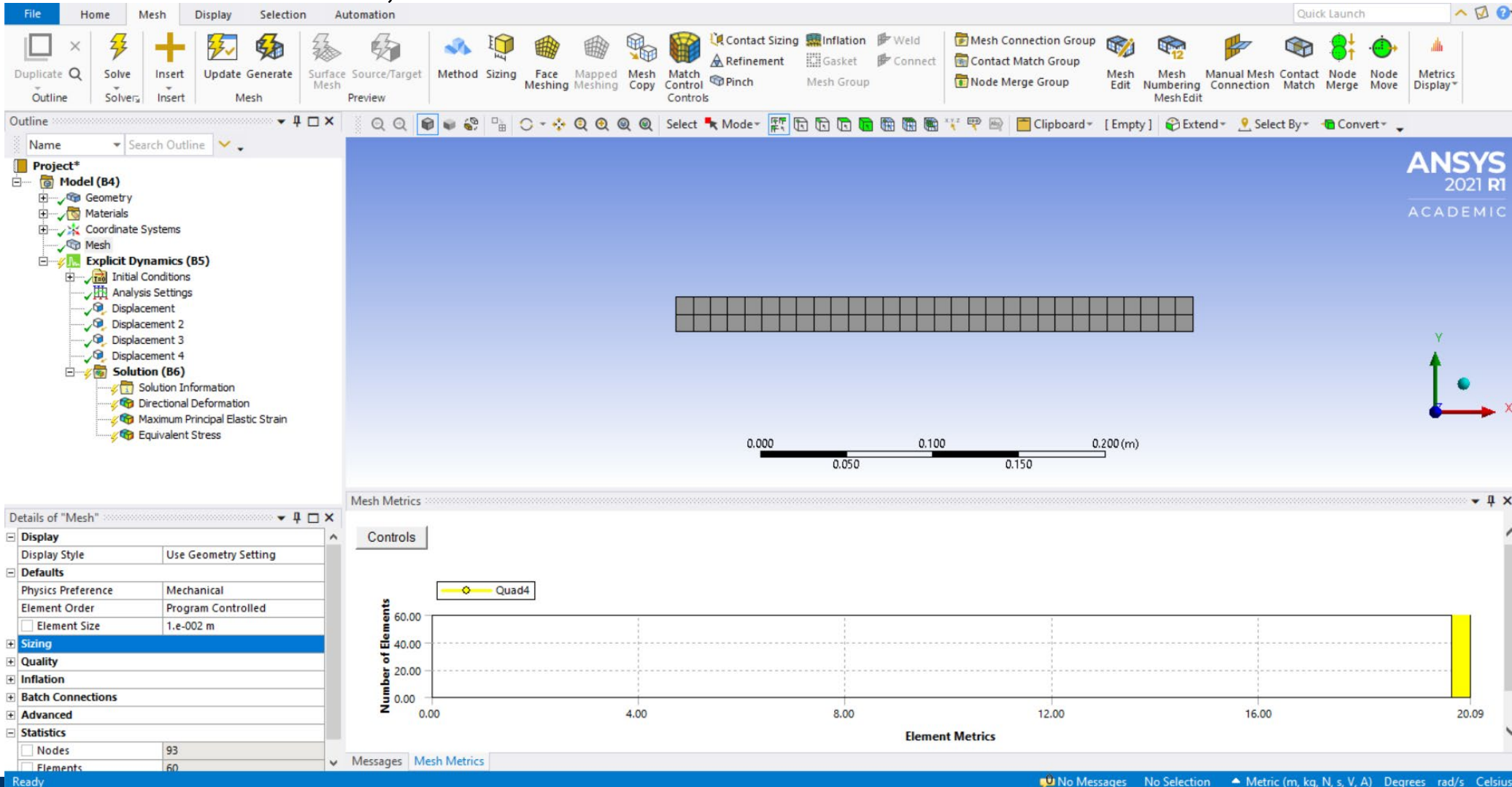
Details of "Structural Steel"

Common Material Properties	
Density	7850 kg/m <sup>3</sup>
Young's Modulus	2e+11 Pa
Thermal Conductivity	60.5 W/m·°C
Specific Heat	434 J/kg·°C
Tensile Yield Strength	2.5e+08 Pa
Tensile Ultimate Strength	4.6e+08 Pa
Nonlinear Behavior	False
Full Details	Click To View Full Details



# • Modelo

$2 \times 30 = 60$  elementos,  $31 \times 3 = 93$  nodos



The screenshot displays the ANSYS 2021 R1 ACADEMIC software interface. The main workspace shows a 2D model of a bar with a mesh of 60 elements and 93 nodes. The bar is oriented horizontally along the X-axis, with a length of 0.200 m. The mesh consists of 30 elements, each 0.0067 m long, and 31 nodes, with 3 nodes per element. The bar is supported at the left end (X=0) and has a force applied at the right end (X=0.200 m).

The **Mesh Metrics** panel at the bottom shows the following data:

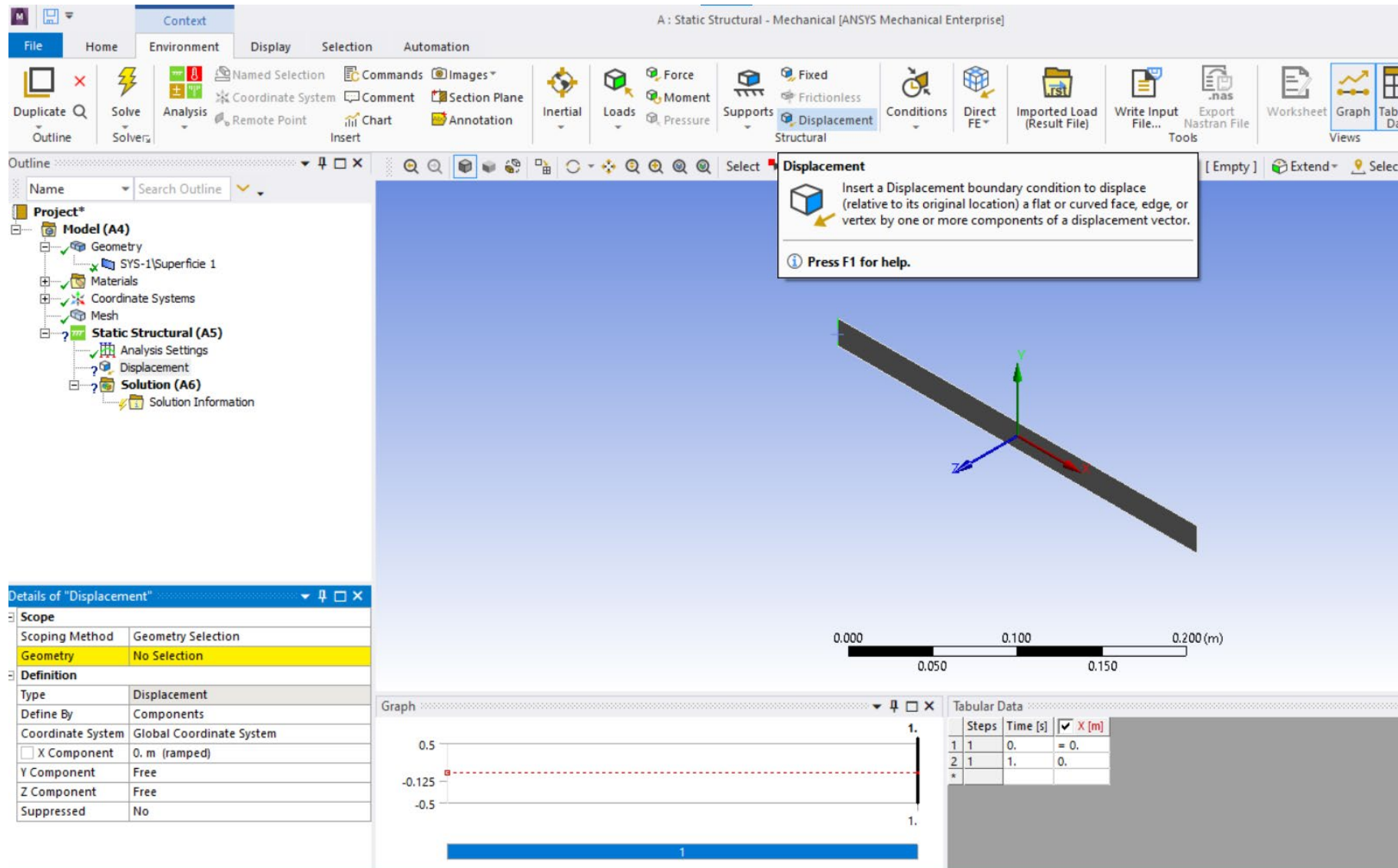
Element Order	Number of Elements
1	30
2	30
3	0
4	0
5	0
6	0
7	0
8	0
9	0
10	0
11	0
12	0
13	0
14	0
15	0
16	0
17	0
18	0
19	0
20	0
21	0
22	0
23	0
24	0
25	0
26	0
27	0
28	0
29	0
30	0
31	0
32	0
33	0
34	0
35	0
36	0
37	0
38	0
39	0
40	0
41	0
42	0
43	0
44	0
45	0
46	0
47	0
48	0
49	0
50	0
51	0
52	0
53	0
54	0
55	0
56	0
57	0
58	0
59	0
60	0

The **Details of "Mesh"** panel shows the following settings:

Category	Value
Display	Use Geometry Setting
Defaults	Physics Preference: Mechanical
Element Order	Program Controlled
Element Size	1.e-002 m
Sizing	
Quality	
Inflation	
Batch Connections	
Advanced	
Statistics	
Nodes	93
Elements	60

# • Modelo

## Editar



The screenshot displays the ANSYS Mechanical Enterprise software interface. The main window shows a 3D model of a beam with a coordinate system (X, Y, Z) and a displacement boundary condition applied. The displacement is defined as 0.0 m in the X direction (ramped) and free in the Y and Z directions. The software interface includes a menu bar (File, Home, Environment, Display, Selection, Automation), a toolbar with various analysis tools, and a detailed view of the displacement boundary condition.

**Displacement**  
Insert a Displacement boundary condition to displace (relative to its original location) a flat or curved face, edge, or vertex by one or more components of a displacement vector.  
Press F1 for help.

**Details of "Displacement"**

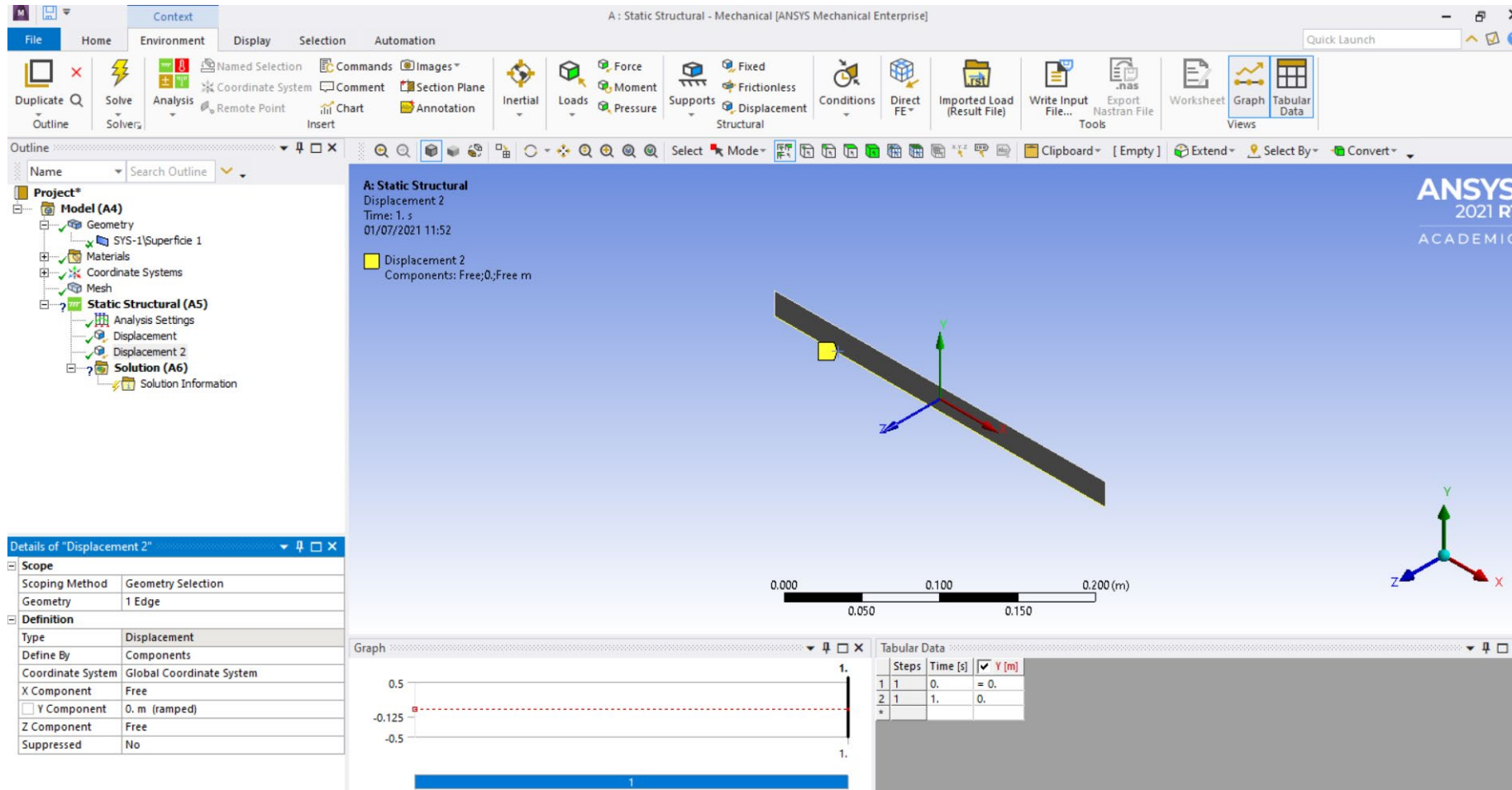
Scope	
Scoping Method	Geometry Selection
Geometry	No Selection
Definition	
Type	Displacement
Define By	Components
Coordinate System	Global Coordinate System
<input type="checkbox"/> X Component	0. m (ramped)
Y Component	Free
Z Component	Free
Suppressed	No

**Graph**

Steps	Time [s]	X [m]
1	0.	= 0.
2	1.	0.

# • Modelo

## Editar



**A: Static Structural**  
Displacement 2  
Time: 1. s  
01/07/2021 11:52

Displacement 2  
Components: Free;0;Free m

**Details of "Displacement 2"**

Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Type	Displacement
Define By	Components
Coordinate System	Global Coordinate System
X Component	Free
<input type="checkbox"/> Y Component	0. m (ramped)
Z Component	Free
Suppressed	No

**Graph**

Steps	Time [s]	Y [m]
1	0.	= 0.
2	1.	0.

# • Modelo

## Editar

A : Static Structural - Mechanical [ANSYS Mechanical Enterprise]

File Home Environment Display Selection Automation

Quick Launch

Duplicate Solve Analysis Named Selection Coordinate System Remote Point Commands Comment Section Plane Images

Force Moment Pressure Supports Fixed Frictionless Displacement Structural Conditions Direct FE Imported Load (Result File) Write Input File... Export Nastran File Tools Worksheet Graph Tabular Data Views

Outline

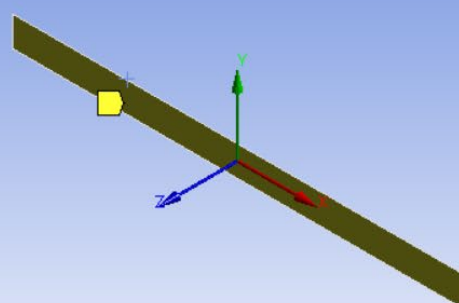
Name Search Outline

Project\*

- Model (A4)
  - Geometry
    - SYS-1\Superficie 1
  - Materials
  - Coordinate Systems
  - Mesh
  - Static Structural (A5)
    - Analysis Settings
    - Displacement
    - Displacement 2
    - Displacement 3
  - Solution (A6)
    - Solution Information

**A: Static Structural**  
Displacement 3  
Time: 1. s  
01/07/2021 11:52

Displacement 3  
Components: Free;Free;0. m



0.000 0.100 0.200 (m)  
0.050 0.150

Details of "Displacement 3"


Scope

Scoping Method	Geometry Selection
Geometry	1 Face

Definition

Type	Displacement
Define By	Components
Coordinate System	Global Coordinate System
X Component	Free
Y Component	Free
Z Component	0. m (ramped)
Suppressed	No

Graph



Tabular Data

Steps	Time [s]	Z [m]
1	0.	= 0.
2	1.	0.
*		

# • Modelo

## Editar

A : Static Structural - Mechanical [ANSYS Mechanical Enterprise]

File Home Environment Display Selection Automation

Quick Launch

Duplicate Outline Solve Analysis Coordinate System Remote Point Insert Chart Annotation Inertial Loads Force Moment Pressure Supports Frictionless Displacement Structural Conditions Direct FE Imported Load (Result File) Write Input File... Export Nastran File Tools Worksheet Graph Tabular Data Views

Outline

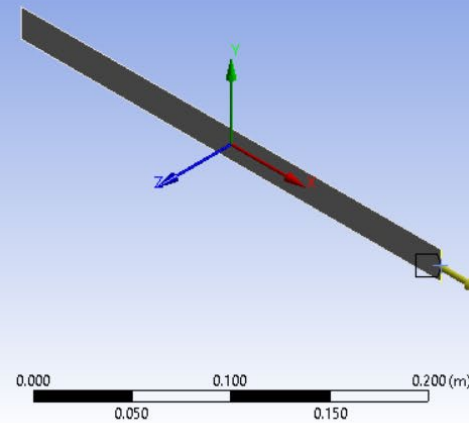
Name Search Outline

Project\*

- Model (A4)
  - Geometry
    - SYS-1\Superficie 1
  - Materials
  - Coordinate Systems
  - Mesh
  - Static Structural (A5)
    - Analysis Settings
    - Displacement
      - Displacement 2
      - Displacement 3
      - Displacement 4
    - Solution (A6)
      - Solution Information

A: Static Structural  
 Displacement 4  
 Time: 1. s  
 01/07/2021 11:59

Displacement 4  
 Components: 1.e-003;Free;Free m



Details of "Displacement 4"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Type	Displacement
Define By	Components
Coordinate System	Global Coordinate System
<input type="checkbox"/> X Component	1.e-003 m (ramped)
Y Component	Free
Z Component	Free
Suppressed	No

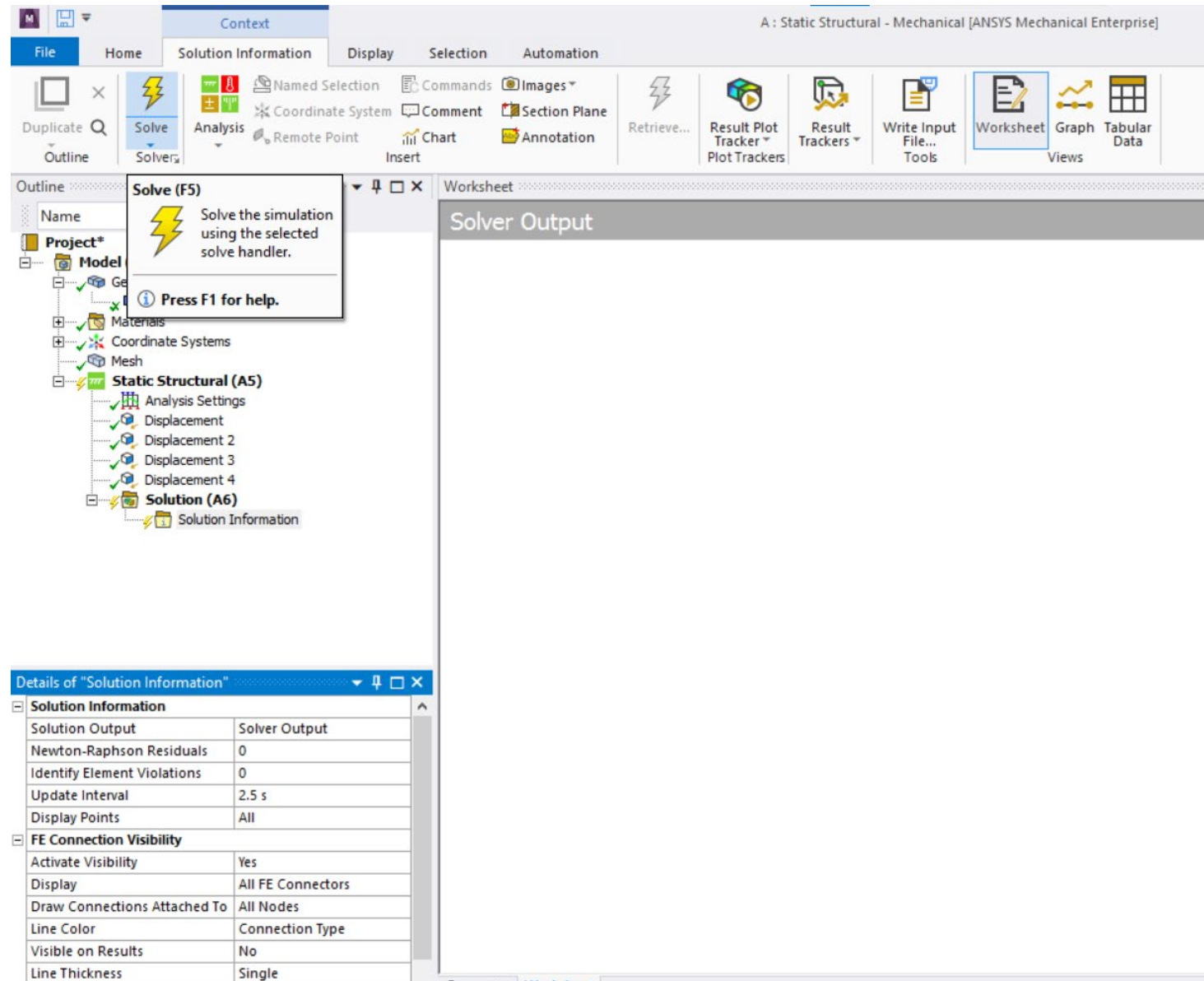


Tabular Data

Steps	Time [s]	X [m]
1	0.	= 0.
2	1.	1.e-003
*		

# • Modelo

## Editar



The screenshot displays the ANSYS Mechanical Enterprise software interface. The top ribbon includes tabs for File, Home, Solution Information, Display, Selection, and Automation. The 'Solution Information' tab is active, showing various tool icons like 'Solve', 'Analysis', 'Coordinate System', 'Remote Point', 'Chart', 'Section Plane', 'Annotation', 'Retrieve...', 'Result Plot Tracker', 'Result Trackers', 'Write Input File...', 'Worksheet', 'Graph', and 'Tabular Data'.

The 'Outline' pane on the left shows a hierarchical tree structure:

- Project\*
- Model
  - Geometric
  - Materials
  - Coordinate Systems
  - Mesh
  - Static Structural (A5)
    - Analysis Settings
    - Displacement
    - Displacement 2
    - Displacement 3
    - Displacement 4
    - Solution (A6)
      - Solution Information

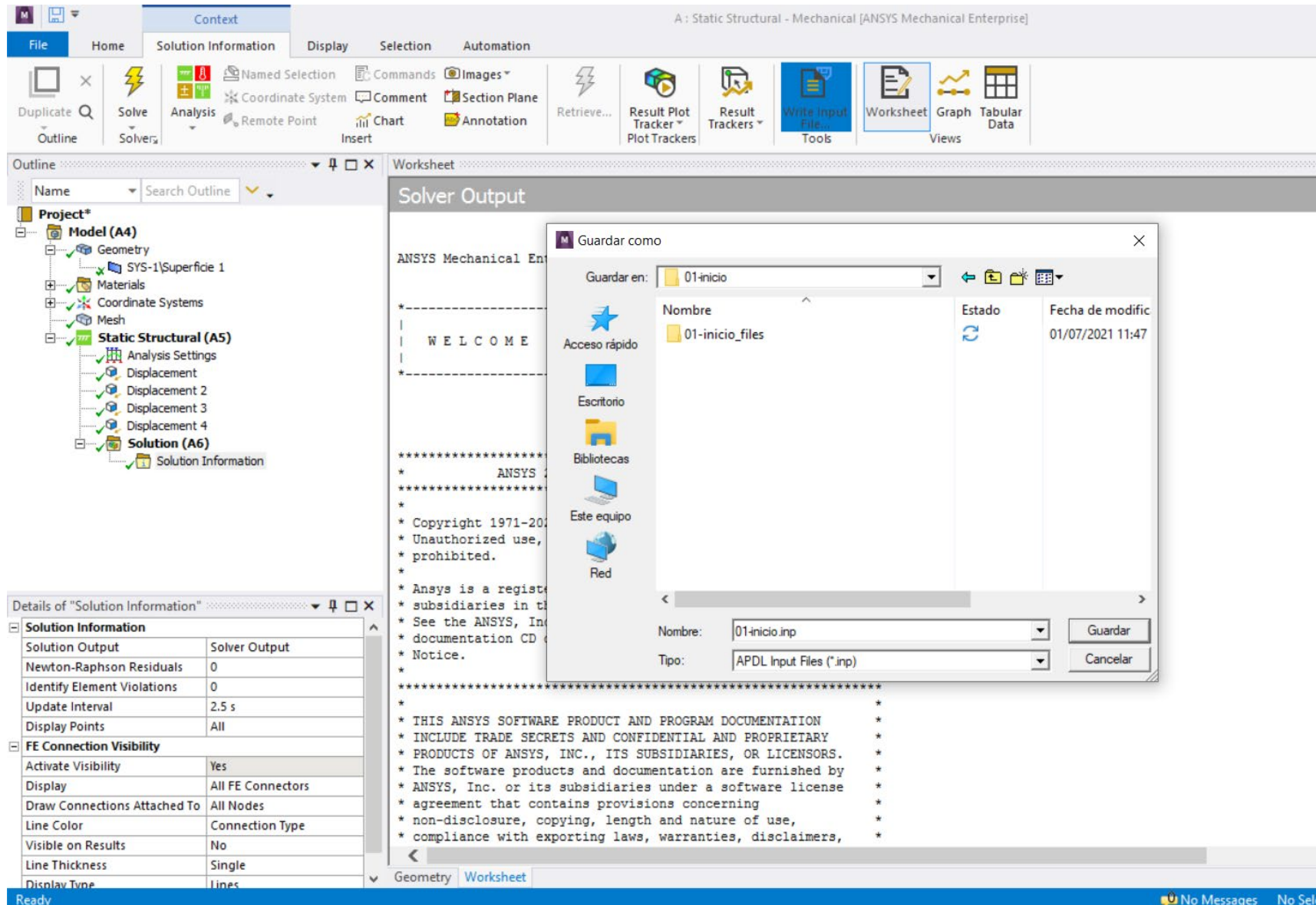
A tooltip for the 'Solve (F5)' button is visible, containing the text: "Solve the simulation using the selected solve handler." and "Press F1 for help."

The 'Details of "Solution Information"' panel at the bottom shows the following data:

Solution Information	
Solution Output	Solver Output
Newton-Raphson Residuals	0
Identify Element Violations	0
Update Interval	2.5 s
Display Points	All
FE Connection Visibility	
Activate Visibility	Yes
Display	All FE Connectors
Draw Connections Attached To	All Nodes
Line Color	Connection Type
Visible on Results	No
Line Thickness	Single

# • Modelo

## Editar



The screenshot displays the ANSYS Mechanical Enterprise software interface. The main window shows a static structural model with a tree view on the left containing: Project\*, Model (A4), Geometry, SYS-1(Superficie 1), Materials, Coordinate Systems, Mesh, Static Structural (A5), Analysis Settings, Displacement, Displacement 2, Displacement 3, Displacement 4, Solution (A6), and Solution Information. The top ribbon includes tabs for File, Home, Solution Information, Display, Selection, and Automation, with various tool icons for solving and analysis. A 'Solver Output' window is open, showing the text 'ANSYS Mechanical Enterprise WELCOME' and a copyright notice. A 'Guardar como' (Save As) dialog box is overlaid on the Solver Output window, with 'Guardar en:' set to '01-inicio', 'Nombre:' set to '01-inicio.inp', and 'Tipo:' set to 'APDL Input Files (\*.inp)'. The dialog also shows a file list with columns for 'Nombre', 'Estado', and 'Fecha de modific'.

ANSYS Mechanical Enterprise

WELCOME

\*\*\*\*\*

ANSYS

\*\*\*\*\*

\* Copyright 1971-2017

\* Unauthorized use,

\* prohibited.

\* Ansys is a registe

\* subsidiaries in th

\* See the ANSYS, Inc

\* documentation CD

\* Notice.

\*\*\*\*\*

\* THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION

\* INCLUDE TRADE SECRETS AND CONFIDENTIAL AND PROPRIETARY

\* PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS.

\* The software products and documentation are furnished by

\* ANSYS, Inc. or its subsidiaries under a software license

\* agreement that contains provisions concerning

\* non-disclosure, copying, length and nature of use,

\* compliance with exporting laws, warranties, disclaimers,

\*\*\*\*\*

Details of "Solution Information"

Solution Information	
Solution Output	Solver Output
Newton-Raphson Residuals	0
Identify Element Violations	0
Update Interval	2.5 s
Display Points	All
FE Connection Visibility	
Activate Visibility	Yes
Display	All FE Connectors
Draw Connections Attached To	All Nodes
Line Color	Connection Type
Visible on Results	No
Line Thickness	Single
Display Type	Lines

# • Editar 01-inicio.inp

```

1 /batch
2 /config,noelddb,1      ! force off writing results to database
3 *get,_wallstrt,active,,time,wall
4 ! ANSYS input file written by Workbench version 2021 R1
5 ! File used for geometry attach: C:\Users\andres.garcia\OneDrive - IQS\ASIGNATURAS\ANSYS\01-inicio\01-inicio_files\dp0\SYS-1\DM\SYS-1.scdoc
6 /title,01-inicio--Static Structural (A5)
7 ! ***** Begin Custom Load Command Snippet *****
8 /COM, ACT Extensions:
9 /COM,      LSDYNA, 2021.1
10 /COM,      5f463412-bd3e-484b-87e7-cbc0a665e474, wbex
11 /COM,
12 ! ***** End Custom Load Command Snippet *****
13 *DIM,_wb_ProjectScratch_dir,string,248
14 _wb_ProjectScratch_dir(1) = 'C:\Users\andres.garcia\OneDrive - IQS\ASIGNATURAS\ANSYS\01-inicio\01-inicio_files\dp0\SYS-1\MECH\'
15 *DIM,_wb_SolverFiles_dir,string,248
16 _wb_SolverFiles_dir(1) = 'C:\Users\andres.garcia\OneDrive - IQS\ASIGNATURAS\ANSYS\01-inicio\01-inicio_files\dp0\SYS-1\MECH\'
17 *DIM,_wb_userfiles_dir,string,248
18 _wb_userfiles_dir(1) = 'C:\Users\andres.garcia\OneDrive - IQS\ASIGNATURAS\ANSYS\01-inicio\01-inicio_files\user_files\'
19 /com,--- Data in consistent MKS units. See Solving Units in the help system for more information.
20 /units,MKS
21 /nopr
22 /wb,file,start      ! signify a WB generated input file
23 /prep7
24 ! Turn off shape checking because checks already performed inside WB mesher.
25 ! See help system for more information.
26 SHPP,OFF,,NOWARN
27 /nolist
28 etcon,set          ! allow ANSYS to choose best KEYOP's for 180x elements, resets any applicable keyopt to MAPDL defaults
29 /com,***** Nodes for the whole assembly *****
30 nblock,3,,105
31 (1i9,3e20.9e3)
32      1      1.500000000E-01      -1.000000000E-02      0.000000000E+00
33      2      1.500000000E-01      1.000000000E-02      0.000000000E+00
34      3      1.500000000E-01      0.000000000E+00      0.000000000E+00
35
36      95      1.306438228E-01      9.364204967E-09      0.000000000E+00
37      96      1.403216358E-01      5.812327962E-09      0.000000000E+00
38
39 -1
40 /wb,elem,start      ! set before creation of elements
41 /com,***** Elements for Body 1 "SYS-1\Superficie 1" *****
42 et,1,181
43 keyo,1,3,2
44 eblock,19,solid,,62
45 (19i9)
46      1      1      1      1      0      0      0      0      4      0      1      67      63      64      94
47      1      1      1      1      0      0      0      0      4      0      2      67      68      62      63

```



# • Editar 01-inicio.inp

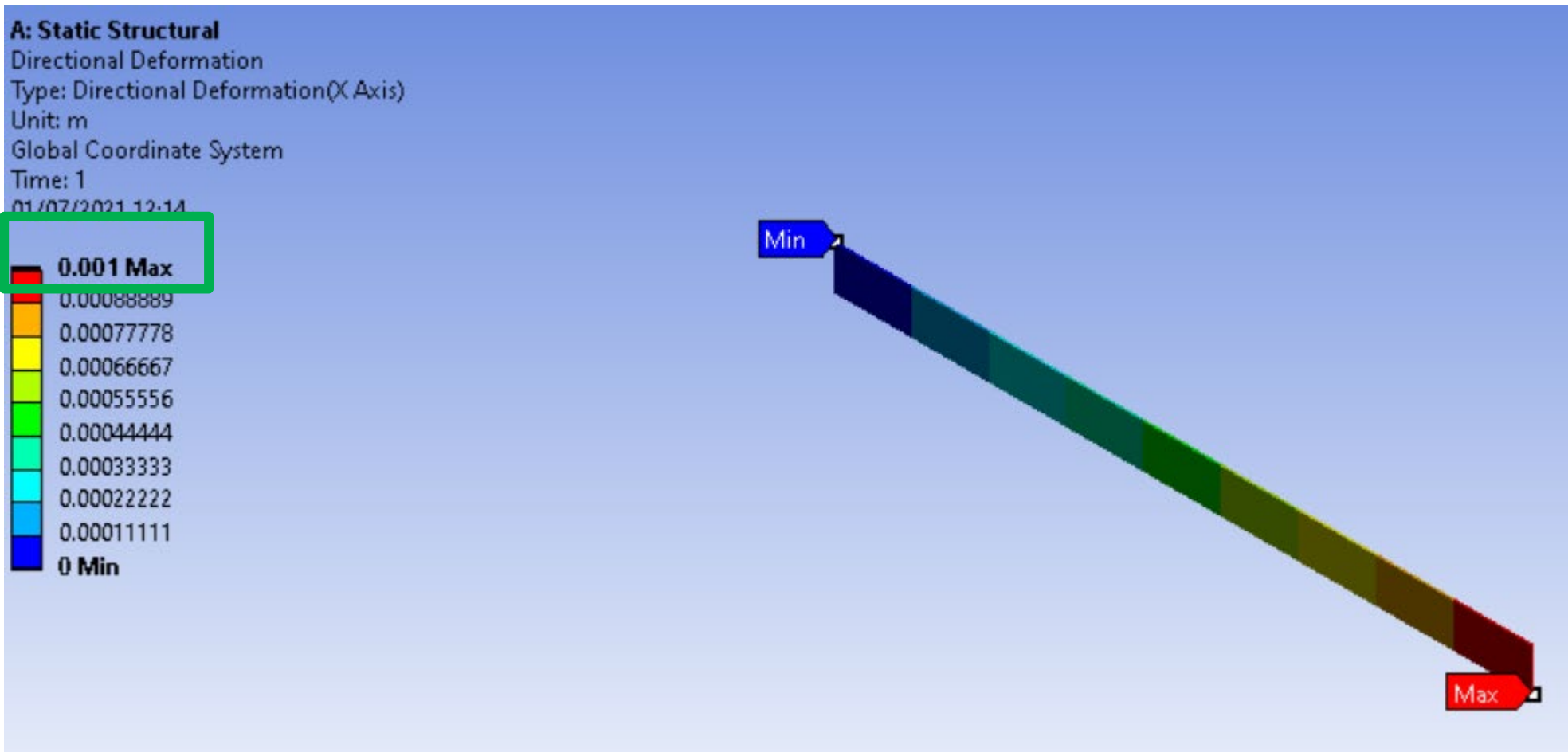
```

195      1      1      1      1      0      0      0      0      4      0      61      66      1      3      96
196      1      1      1      1      0      0      0      0      4      0      62      3      2      5      96
197 -1
198 !Material Id = {B17834B5-0960-437F-8E88-2E143AC17914}
199 /wb,elem,end          ! done creating elements
200 /com,***** Send User Defined Coordinate System(s) *****
201 csys,0
202 tofst,273.15, ! Temperature offset from absolute zero
203 /com,***** Set Reference Temperature *****
204 tref,22.
205 /wb,mat,start        ! starting to send materials
206 /com,***** Send Materials *****
207 Temperature = 'TEMP' ! Temperature
208 MP,DENS,1,7850, ! kg m^-3
209 MP,ALPX,1,1.2e-05, ! C^-1
210 MP,C,1,434, ! J kg^-1 C^-1
211 MP,KXX,1,60.5, ! W m^-1 C^-1
212 MP,RSVX,1,1.7e-07, ! kg m^3 A^-2 s^-3
213 MP,EX,1,200000000000, ! Pa
214 MP,NUXY,1,0.3,
215 MP,MURX,1,10000,
216
217 /wb,mat,end          ! done sending materials
218 /com,***** Send Sheet Properties *****
219 sectype,1,shell
220 secdata,1.e-003
221 secoff,mid
222
223 !***** Model Summary *****
224 !SYS-1\Superficie 1, Structural Steel, matid, 1
225 !***** End Model Summary *****
226 ! get the diagonal of the bounding box. Needed later for other things

```

## • Results

Comprobamos que hemos estirado **1mm en X** como especificado **OK**



## • Results

deformación:  $\varepsilon = \Delta L / L = 1/300 = 0.003333[-] = 0.3333\%$  **OK**

### A: Static Structural

Equivalent Elastic Strain 2

Type: Equivalent Elastic Strain - Top/Bottom

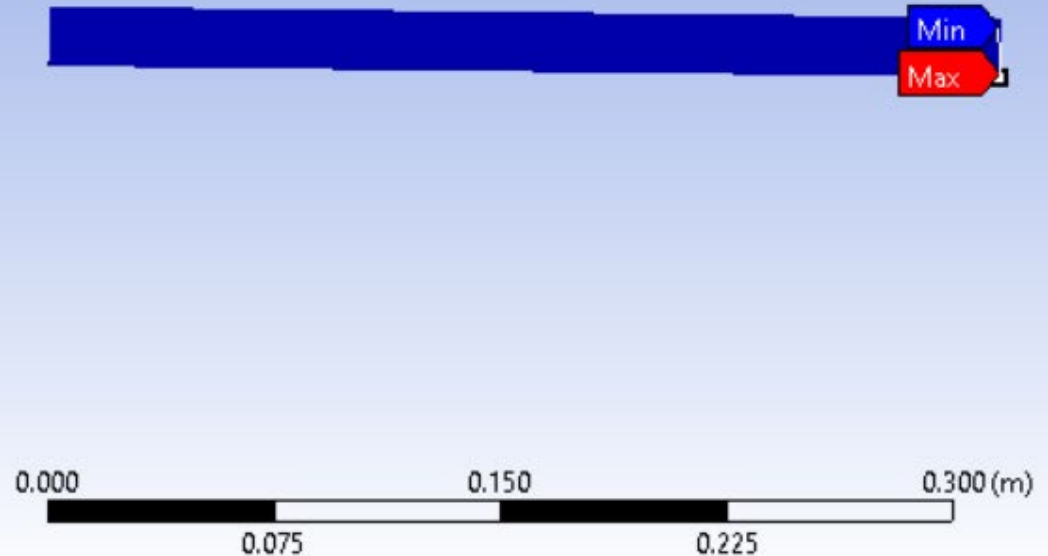
Unit: m/m

Time: 1

01/07/2021 12:20

0.0033333 Max

0.0033333 Min



## • Results

tensión:  $\sigma = Ee = 200[\text{GPa}] \cdot 0.0033 = 0.6667[\text{GPa}] = 666.7[\text{MPa}] = 6.667e8[\text{Pa}]$  OK

### A: Static Structural

Equivalent Stress

Type: Equivalent (von-Mises) Stress - Top/Bottom

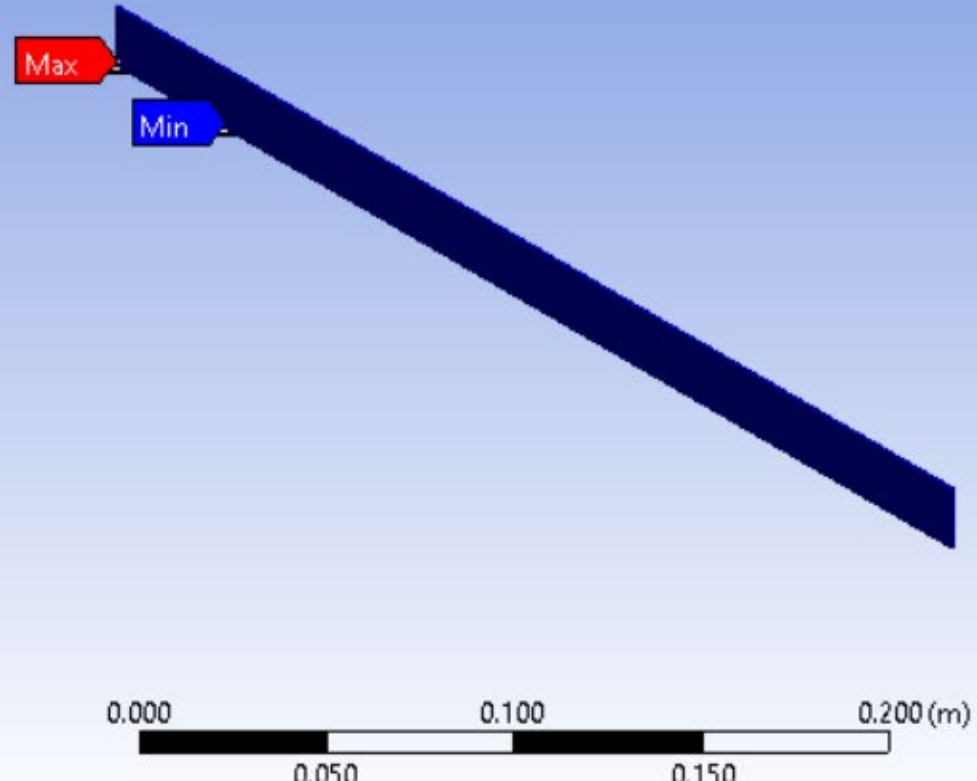
Unit: Pa

Time: 1

01/07/2021 12:16

6.6667e8 Max

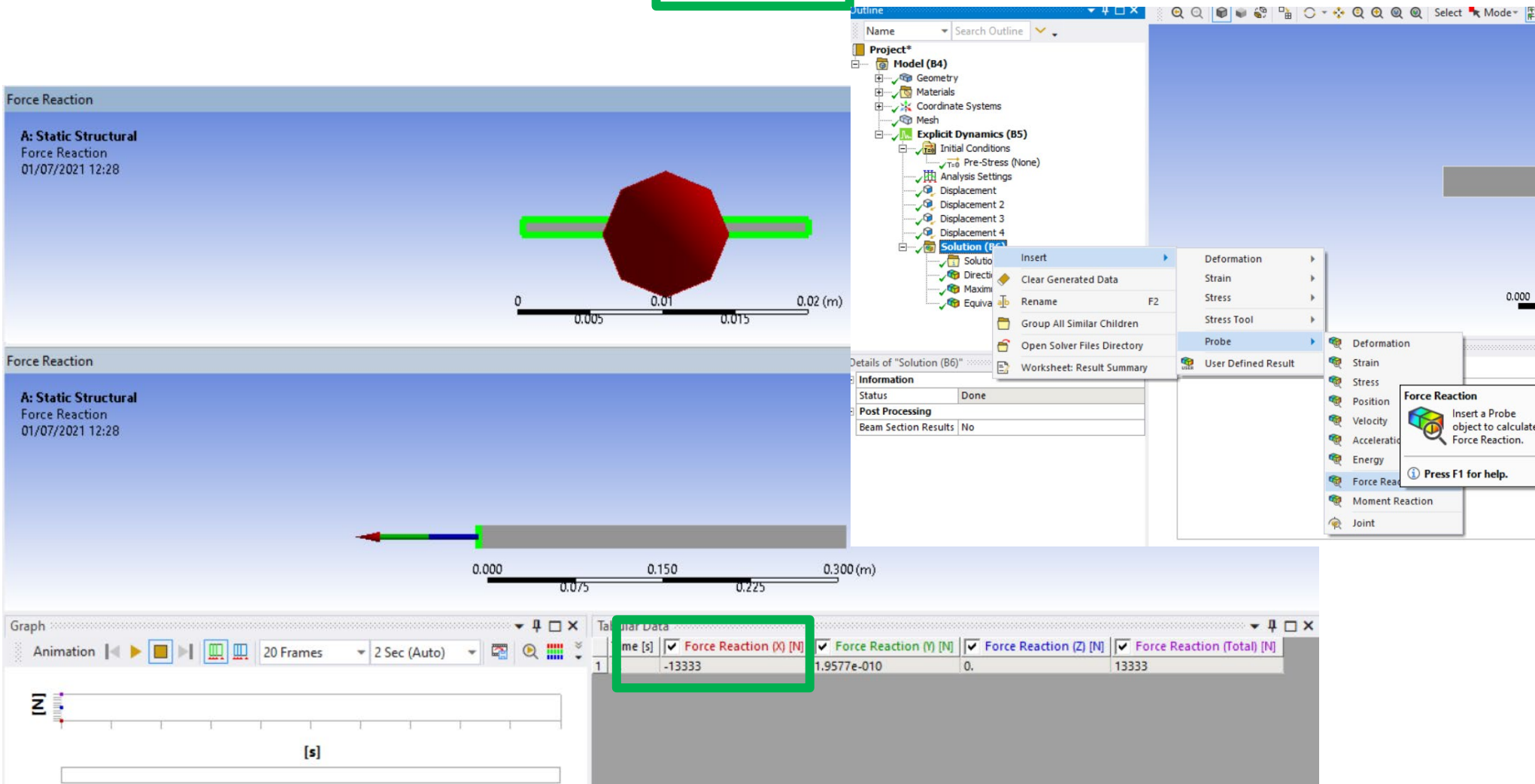
6.6667e8 Min



# • Results

fuerza:

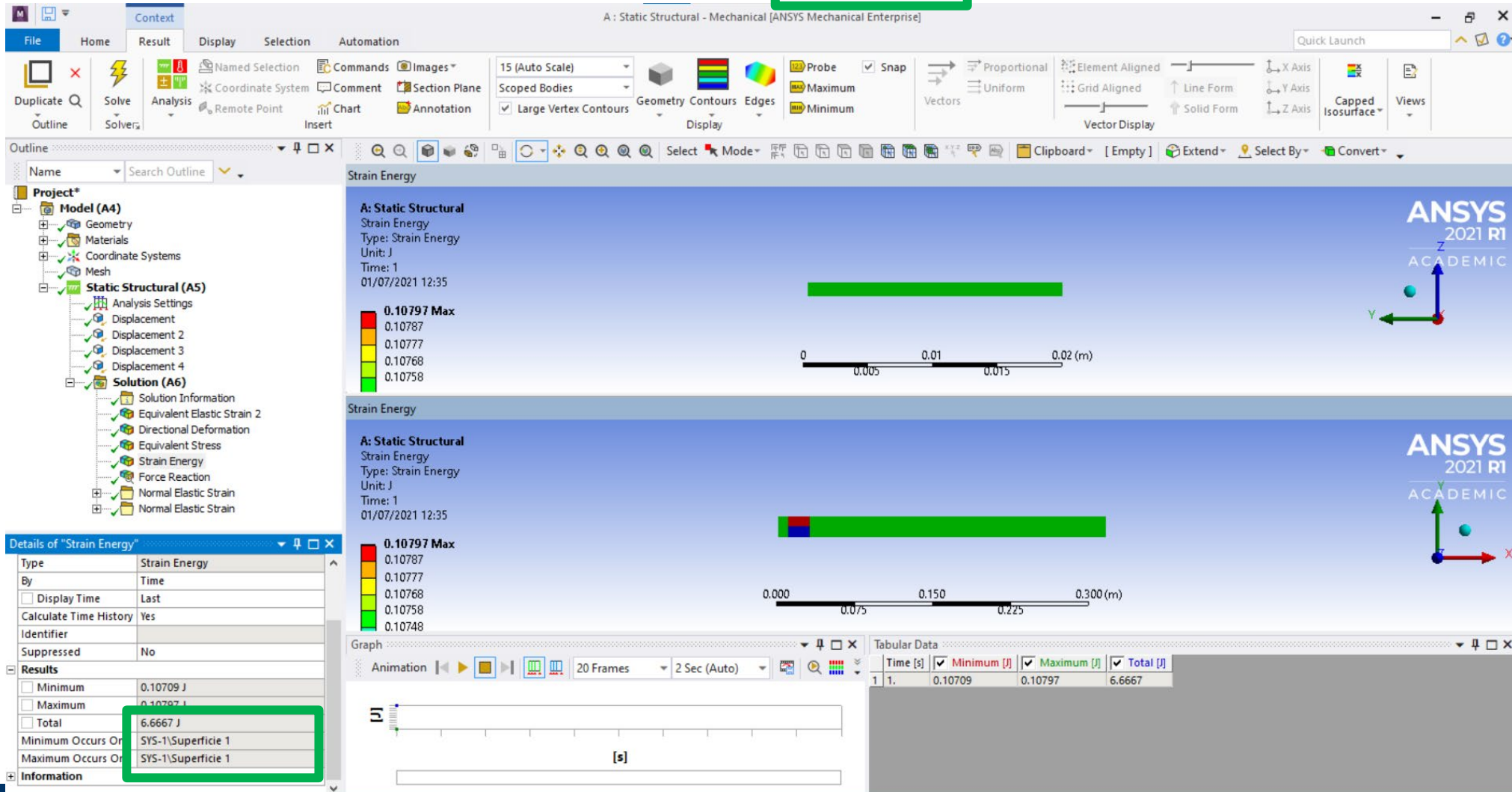
$$F = \sigma A = 666.7 * 20 * 1 = 13333 \text{ [N]} = 13.33 \text{ [kN]} \text{ OK}$$



# • Results

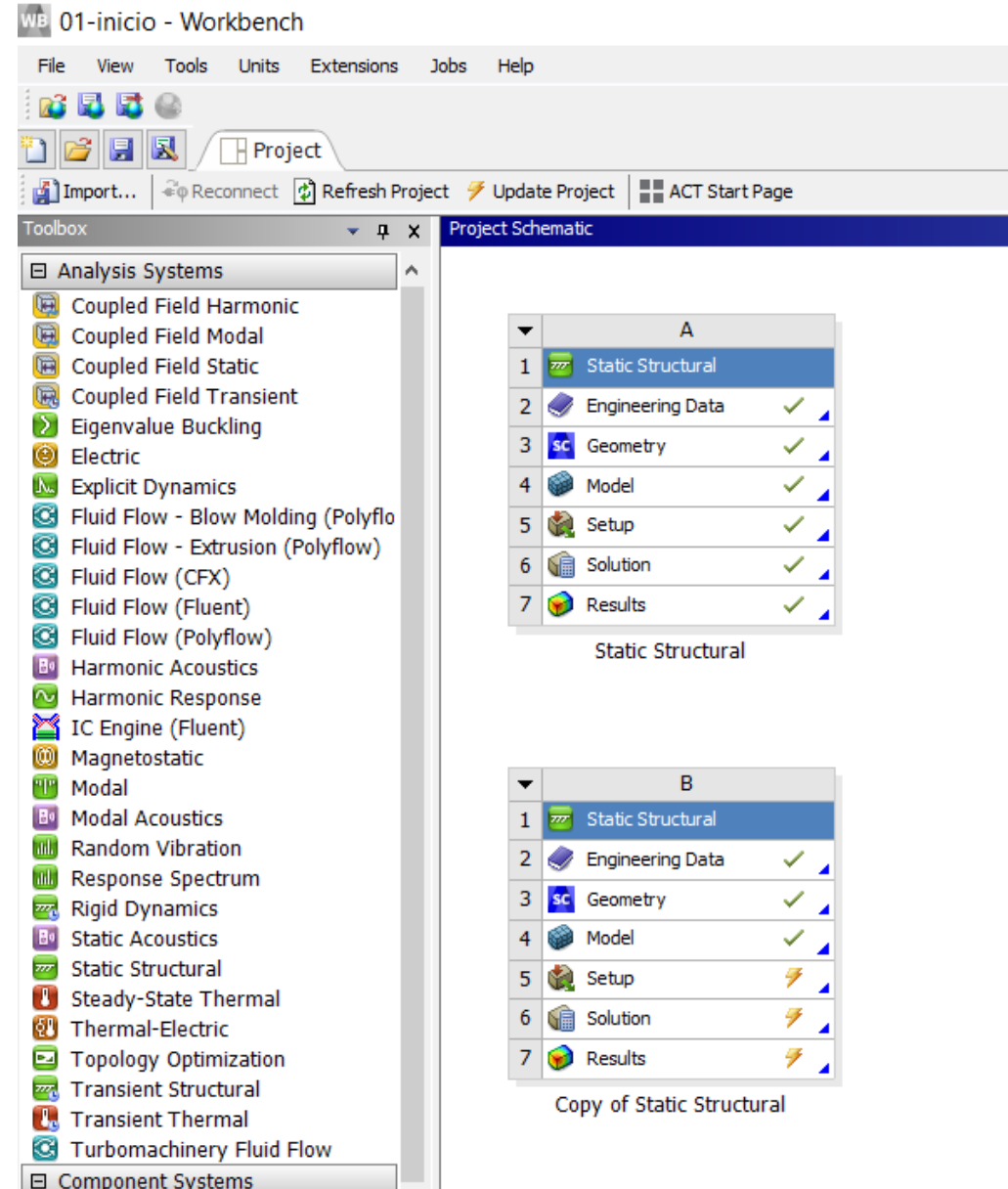
energía:

$$E = F \Delta L / 2 = 13333 * 0.001 / 2 = 6.667 \text{ [J]} = 6.667 \text{ [kN} \cdot \text{mm]} \text{ OK}$$



## • Dinámica

Repetimos el proceso pero haciendo una copia del estático



The screenshot shows the ANSYS Workbench interface for a project named "01-inicio - Workbench". The "Project Schematic" window displays two analysis systems, A and B, both based on a "Static Structural" analysis type.

**Analysis System A: Static Structural**

1	Static Structural	
2	Engineering Data	✓
3	Geometry	✓
4	Model	✓
5	Setup	✓
6	Solution	✓
7	Results	✓

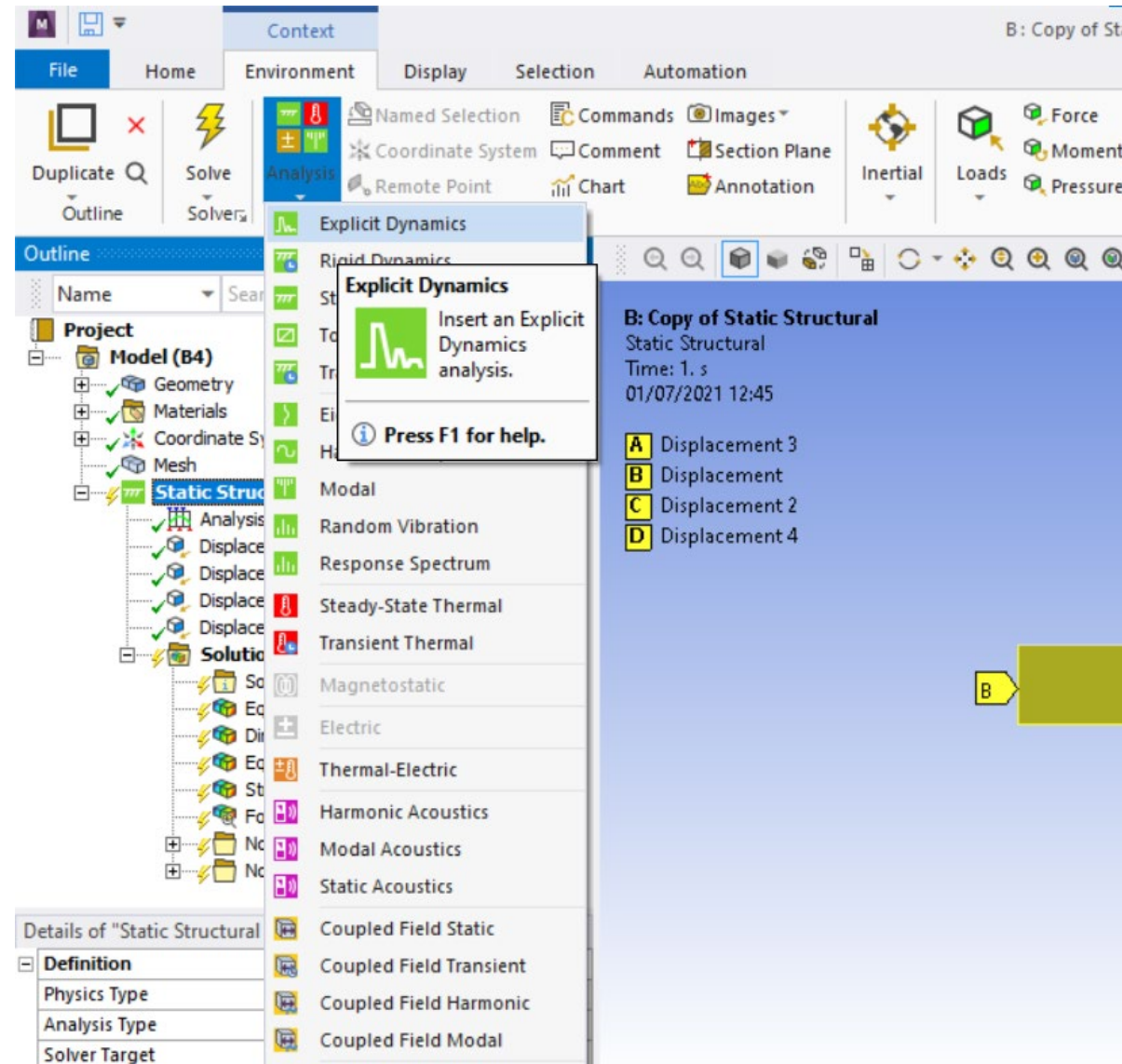
**Analysis System B: Copy of Static Structural**

1	Static Structural	
2	Engineering Data	✓
3	Geometry	✓
4	Model	✓
5	Setup	⚡
6	Solution	⚡
7	Results	⚡

The "Toolbox" on the left lists various analysis systems, including "Static Structural", "Transient Structural", "Harmonic Response", "Eigenvalue Buckling", "Fluid Flow", "Electric", "Magnetostatic", "Modal", "Random Vibration", "Response Spectrum", "Rigid Dynamics", "Static Acoustics", "Steady-State Thermal", "Thermal-Electric", "Topology Optimization", "Transient Thermal", and "Turbomachinery Fluid Flow".

# • Setup

Desde Setup cambiamos a explícito





# • Setup

B: Explicit Dynamics - Mechanical [ANSYS Mechanical Enterprise]

File Home Environment Display Selection Automation

Duplicate Outline Solve Solvers Analysis Coordinate System Remote Point Insert Commands Comment Chart Annotation Images Section Plane

Inertial Loads Force Moment Pressure Supports Fixed Frictionless Displacement Structural Conditions Direct FE Write Input File... Export Nastran File Tools Worksheet Graph Tabular Data Views

Outline Name Search Outline

Project\*

- Model (B4)
  - Geometry
  - Materials
  - Coordinate Systems
  - Mesh
  - Explicit Dynamics (B5)
    - Initial Conditions
    - Pre-Stress (None)
    - Analysis Settings
    - Displacement
    - Displacement 2
    - Displacement 3
    - Displacement 4
  - Solution (B6)
    - Solution Information

Details of "Analysis Settings"

Analysis Settings Preference

Type	Program Controlled
<b>Step Controls</b>	
Number Of Steps	1
Current Step Number	1
Load Step Type	Explicit Time Integration
End Time	1.e-003 s
Resume From Cycle	0
Maximum Number of Cycles	1e+07
Maximum Energy Error	0.1
Reference Energy Cycle	0
Initial Time Step	Program Controlled

Graph

0.000 0.025 0.050 0.075 0.100 (m)

1.e-3 3.75e-4 0. 1.e-3

Tabular Data

Steps	End Time [s]
1	1.e-003
*	

## • Setup

Context: B: Explicit Dynamics - Mechanical [ANSYS Mechanical Enterprise]

File Home Environment Display Selection Automation Quick Launch


Duplicate Outline | Solve Solvers | Analysis | Named Selection | Coordinate System | Remote Point | Comment | Chart | Section Plane | Annotation | Inertial | Loads | Force | Moment | Pressure | Supports | Fixed | Frictionless | Displacement | Conditions | Direct FE | Write Input File... | Export Nastran File | Worksheet | Graph | Tabular Data

Outline: Name Search Outline

- Project\*
  - Model (B4)
    - Geometry
    - Materials
    - Coordinate Systems
    - Mesh
  - Explicit Dynamics (B5)
    - Initial Conditions
    - Pre-Stress (None) T=0
    - Analysis Settings
    - Displacement
    - Displacement 2
    - Displacement 3
    - Displacement 4
  - Solution (B6)
    - Solution Information

**B: Explicit Dynamics**  
 Displacement 4  
 Time: 1.e-003 s  
 01/07/2021 12:52

Displacement 4  
 Components: 1.e-003;Free;Free m



0.000 0.025 0.050 0.075 0.100 (m)

Graph: 1.e-3 0. 1.e-3

Tabular Data

Steps	Time [s]	X [m]
1	0.	0.
2	1.e-003	1.e-003
*		

Details of "Displacement 4"

Scope

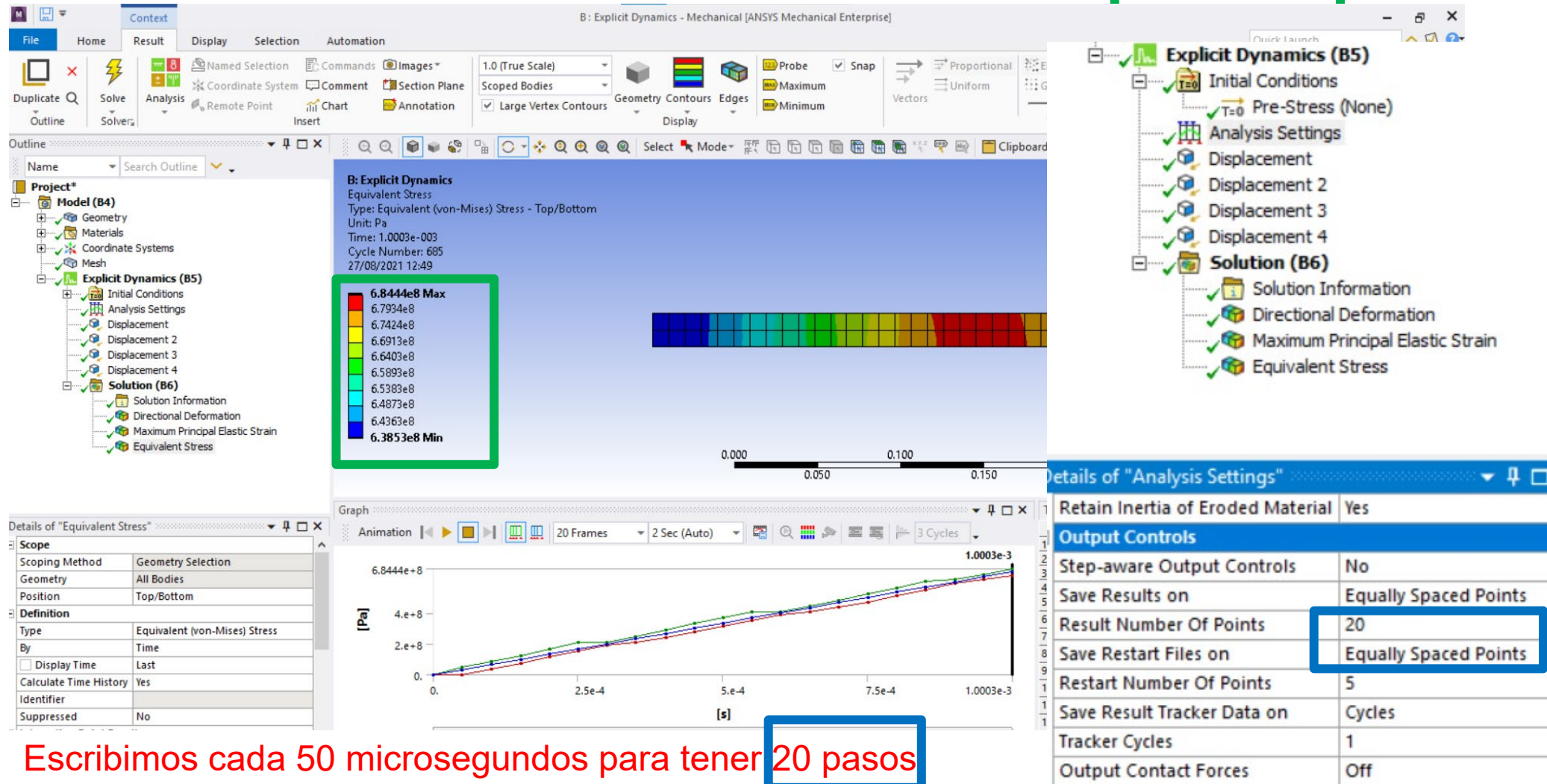
Scoping Method	Geometry Selection
Geometry	1 Edge

Definition

Type	Displacement
Define By	Components
Coordinate System	Global Coordinate System
X Component	1.e-003 m (ramped)
Y Component	Free
Z Component	Free
Suppressed	No

# • Results

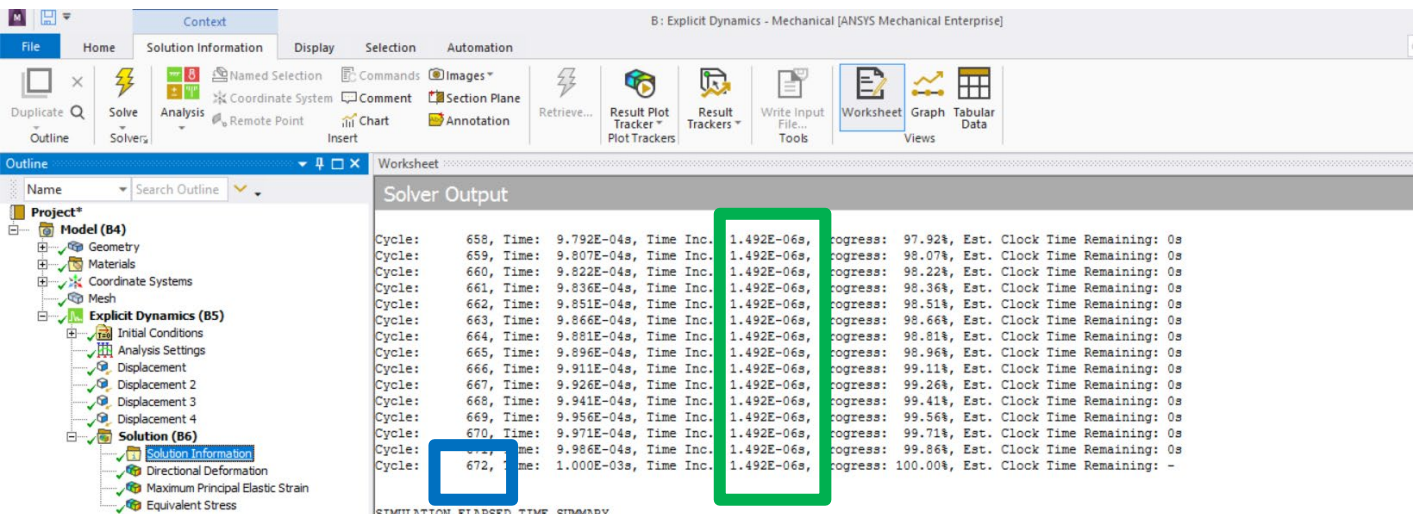
tensión:  $\sigma = Ee = 200[\text{GPa}] \cdot 0.0033 = 0.6667[\text{GPa}] = 666.7[\text{MPa}] = 6.667e8[\text{Pa}]$  OK



Escribimos cada 50 microsegundos para tener 20 pasos

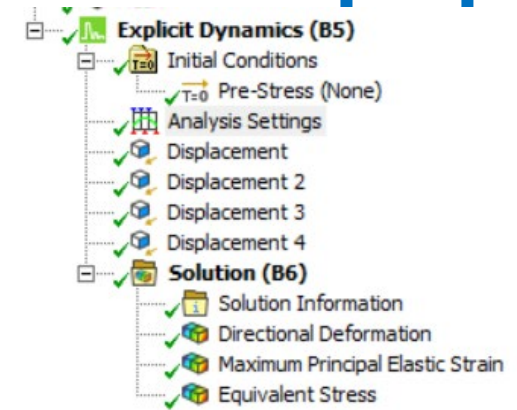
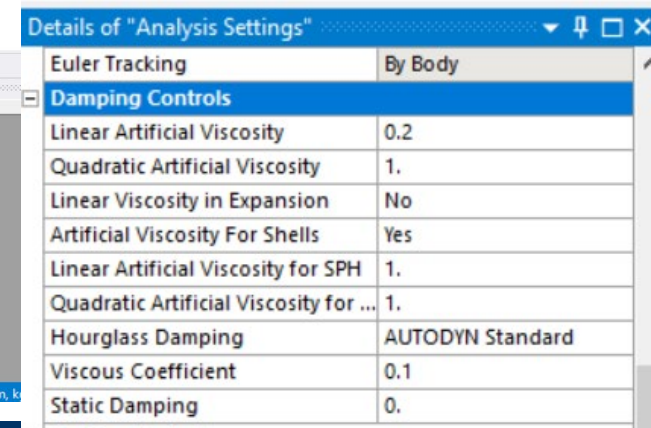
## • Time step

Como tenemos una malla de longitud 10mm de acero el time step debería ser de unos 1.9 microsegundos y algo menos debido al amortiguamiento. Por lo tanto para calcular un milisegundo necesitará  $0.001/1.49e-6 = 671$  pasos mas el inicia 672



Solver Output

Cycle	Time	Time Inc.	Progress	Est. Clock Time Remaining
658	9.792E-04s	1.492E-06s	97.92%	0s
659	9.807E-04s	1.492E-06s	98.07%	0s
660	9.822E-04s	1.492E-06s	98.22%	0s
661	9.836E-04s	1.492E-06s	98.36%	0s
662	9.851E-04s	1.492E-06s	98.51%	0s
663	9.866E-04s	1.492E-06s	98.66%	0s
664	9.881E-04s	1.492E-06s	98.81%	0s
665	9.896E-04s	1.492E-06s	98.96%	0s
666	9.911E-04s	1.492E-06s	99.11%	0s
667	9.926E-04s	1.492E-06s	99.26%	0s
668	9.941E-04s	1.492E-06s	99.41%	0s
669	9.956E-04s	1.492E-06s	99.56%	0s
670	9.971E-04s	1.492E-06s	99.71%	0s
671	9.986E-04s	1.492E-06s	99.86%	0s
672	1.000E-03s	1.492E-06s	100.00%	-

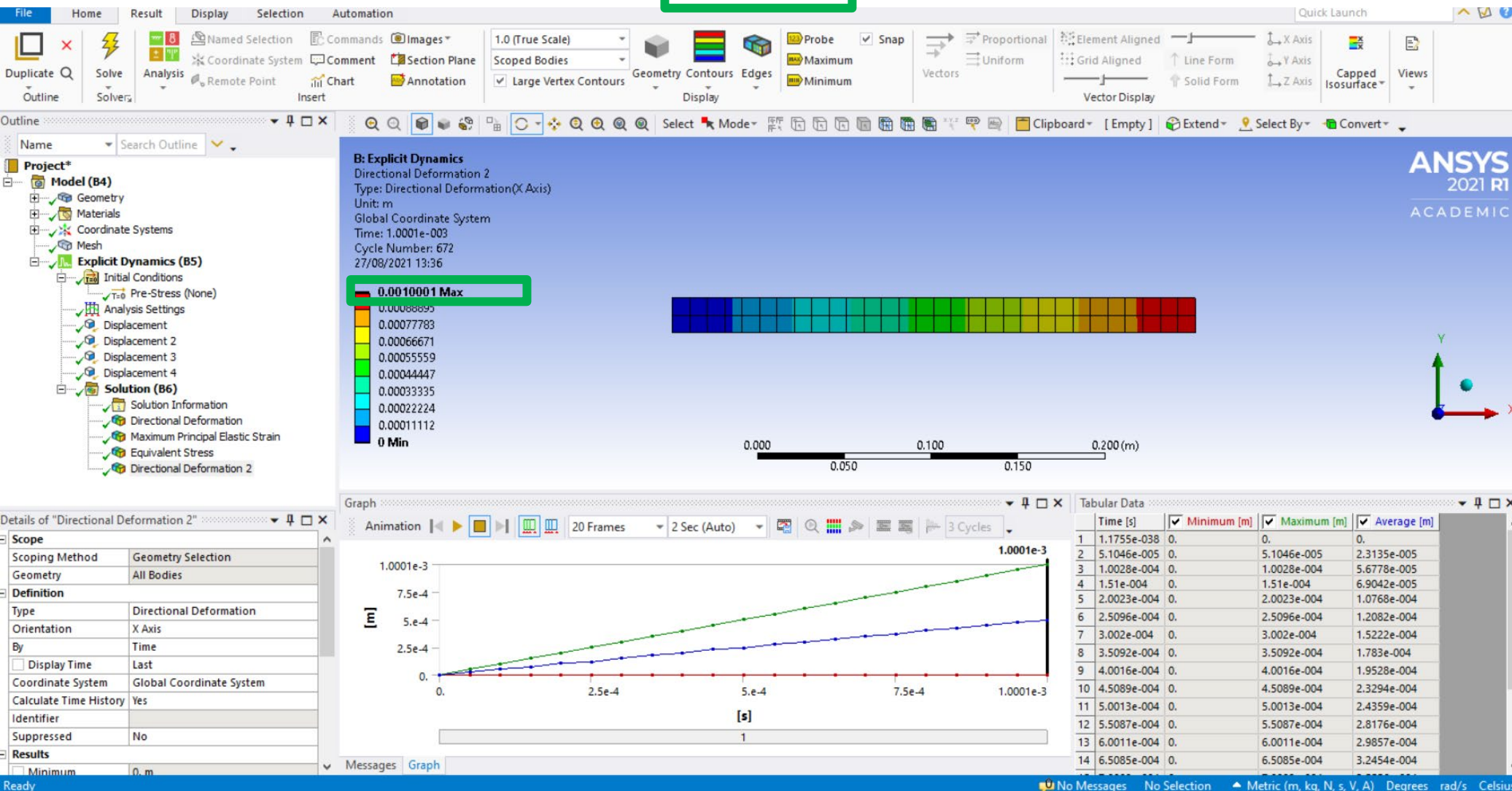



Euler Tracking		By Body
<b>Damping Controls</b>		
Linear Artificial Viscosity	0.2	
Quadratic Artificial Viscosity	1.	
Linear Viscosity in Expansion	No	
Artificial Viscosity For Shells	Yes	
Linear Artificial Viscosity for SPH	1.	
Quadratic Artificial Viscosity for ...	1.	
Hourglass Damping	AUTODYN Standard	
Viscous Coefficient	0.1	
Static Damping	0.	

	[-]	1	0.9
		Steel	Steel
E	[GPa]	210	210
d	[kg/mm <sup>3</sup> ]	7.85E-06	7.85E-06
Time step for L10	[μs]	1.93E+00	1.74E+00
Time step for L5	[μs]	9.67E-01	8.70E-01
L for time 1μs	[mm]	5.17E+00	5.75E+00
L_for_time_0.5μs	[mm]	2.59E+00	2.87E+00

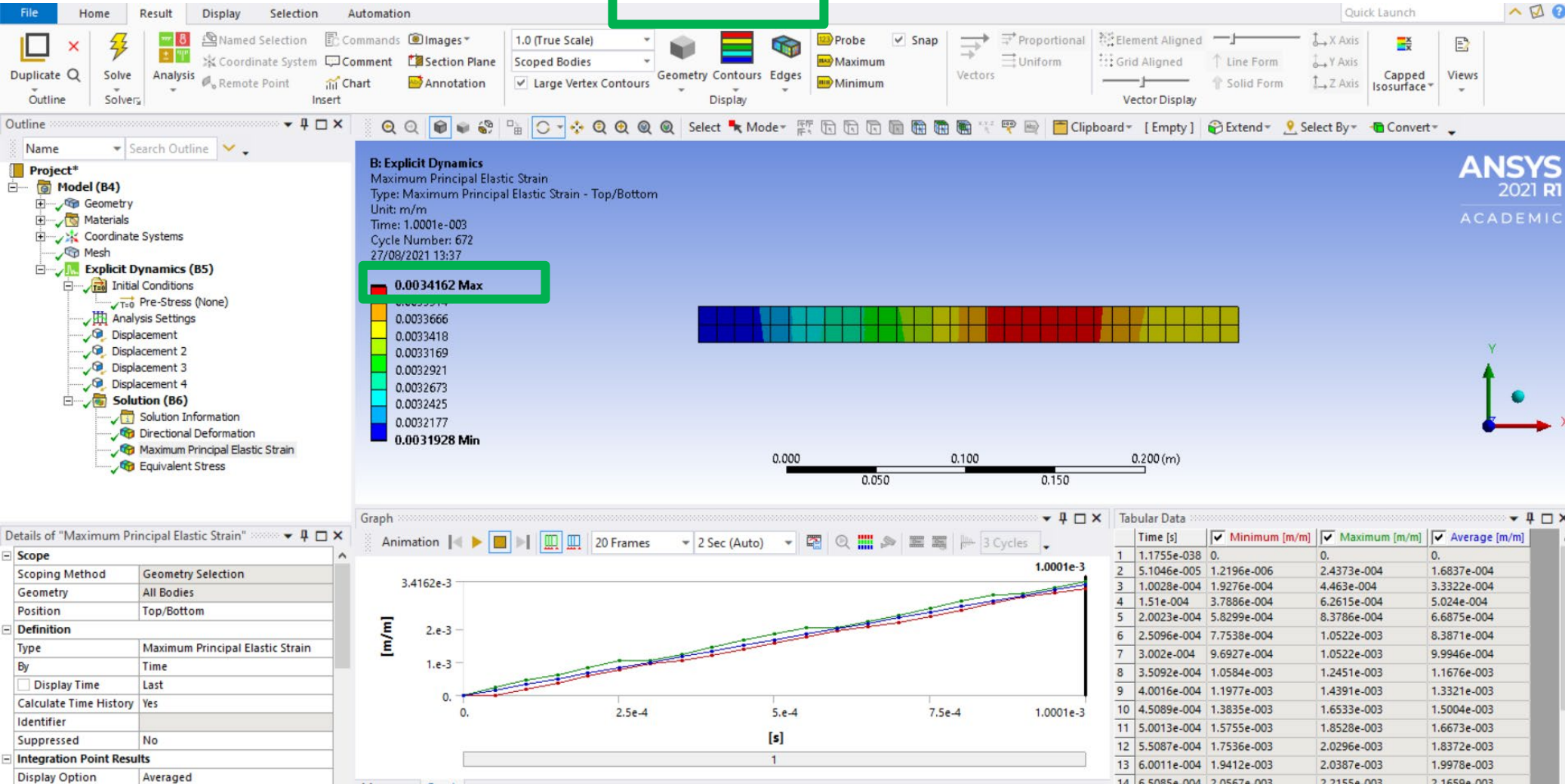
# • Results

Comprobamos que hemos estirado **1mm en X** como especificado **OK**



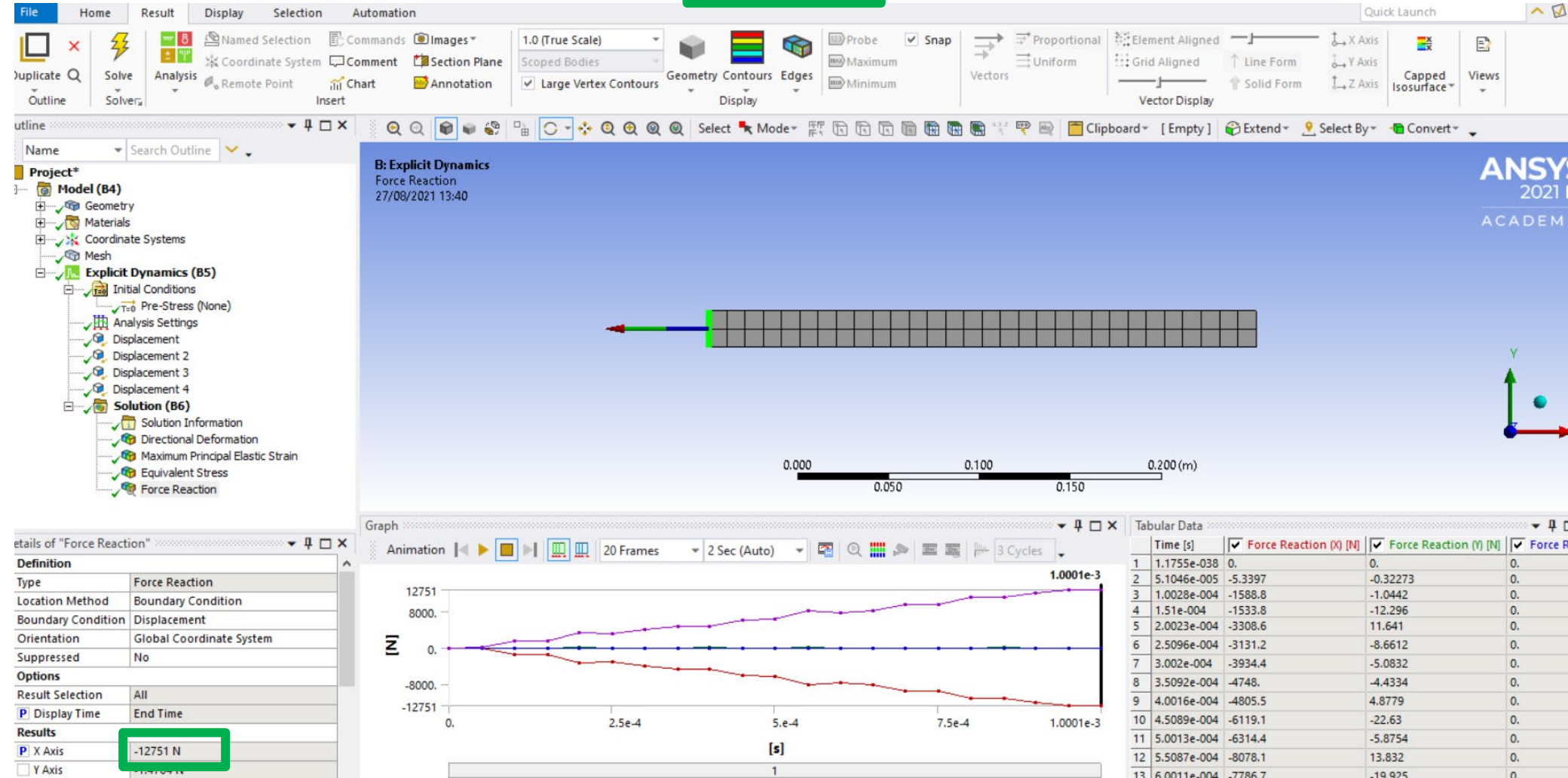
# • Results

deformación:  $\varepsilon = \Delta L / L = 1/300 = 0.003333[-] = 0.3333\%$  **OK**



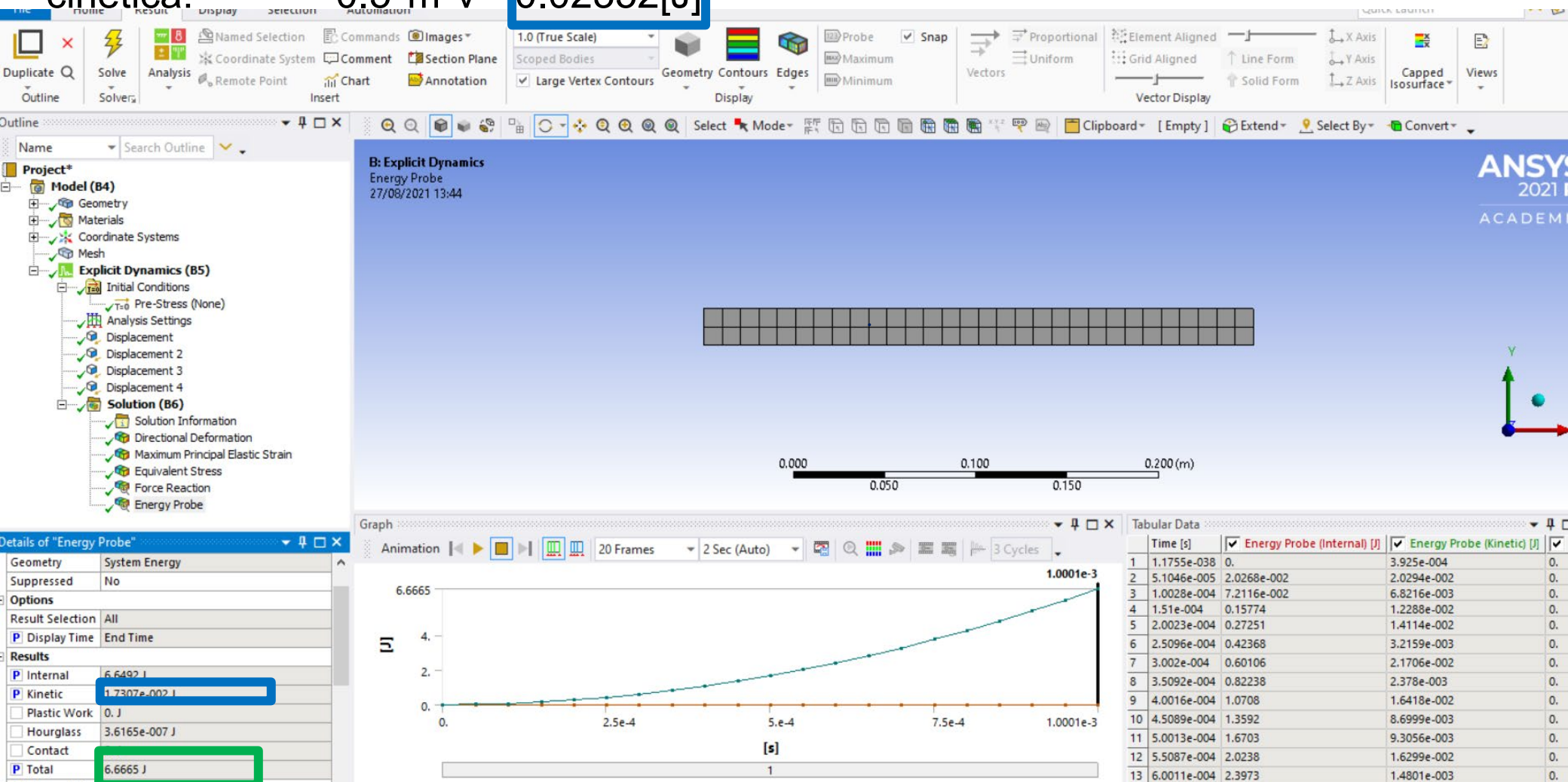
# • Results

fuerza:  $F = \sigma A = 666.7 \cdot 20 \cdot 1 = 13333[\text{N}] = 13.33[\text{kN}]$  OK simulación 12.75[kN]



# • Results

energía:  $E = F \Delta L / 2 = 13333 \cdot 0.001 / 2 = 6.667 \text{ [J]} = 6.667 \text{ [kN} \cdot \text{mm]} \text{ OK} + \text{energía}$   
 cinética:  $0.5 \cdot m \cdot v^2 = 0.02352 \text{ [J]}$





## • Ejercicio

Realizar el mismo cálculo con malla de 2.5mm ¿Cuántos elementos tendremos?  
 ¿Cuánto durará el cálculo? ¿Hay diferencias en los resultados?

Ahora tenemos los 60elementos x 4x4=960. Además el time step es 4 veces más pequeño y por tanto tendremos un tiempo de cálculo x4x4x4=64 pero en este cálculo tan rápido no se nota. La fuerza pasa de 12.75[kN] a 12.81[kN] similar dentro de un cálculo explícito

Cycle: 672, Time: 1.000E-03s, Time Inc.: 1.492E-06s << 01-inicio\_files > dp0 > SYS-4 > MECH

### SIMULATION ELAPSED TIME SUMMARY

EXECUTION FROM CYCLE 0 TO 672  
 ELAPSED RUN TIME IN SOLVER = 3.88333E-03 Minutes  
 TOTAL ELAPSED RUN TIME = 1.36167E-02 Minutes  
 JOB RAN OVER 2 WORKERS  
 JOB RAN USING Intel MPI  
 JOB RAN USING DECOMPOSITION AUTO

Problem terminated .... wrapup time reached

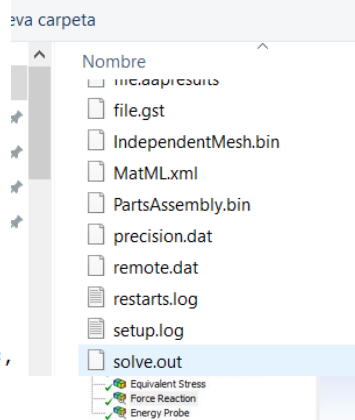


Cycle: 2683, Time: 1.000E-03s, Time Inc.: 3.731E-07s,

### SIMULATION ELAPSED TIME SUMMARY

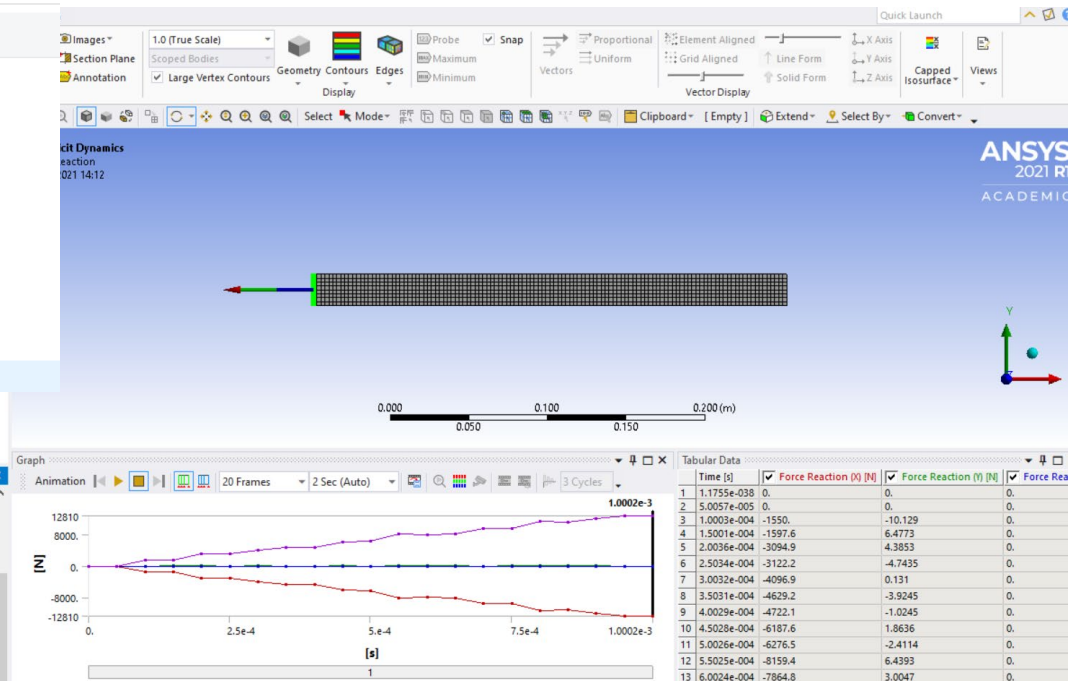
EXECUTION FROM CYCLE 0 TO 2683  
 ELAPSED RUN TIME IN SOLVER = 4.40000E-02 Minutes  
 TOTAL ELAPSED RUN TIME = 6.54667E-02 Minutes  
 JOB RAN OVER 2 WORKERS  
 JOB RAN USING Intel MPI  
 JOB RAN USING DECOMPOSITION AUTO

Problem terminated .... wrapup time reached



Details of "Force Reaction"

Display Time	End Time
Results	
Maximum Value Over Time	
X Axis	0. N
Y Axis	6.4773 N
Z Axis	0. N
Total	12810 N
Minimum Value Over Time	
X Axis	-12810 N
Y Axis	-10.129 N
Z Axis	0. N
Total	0. N



## • Ejercicio

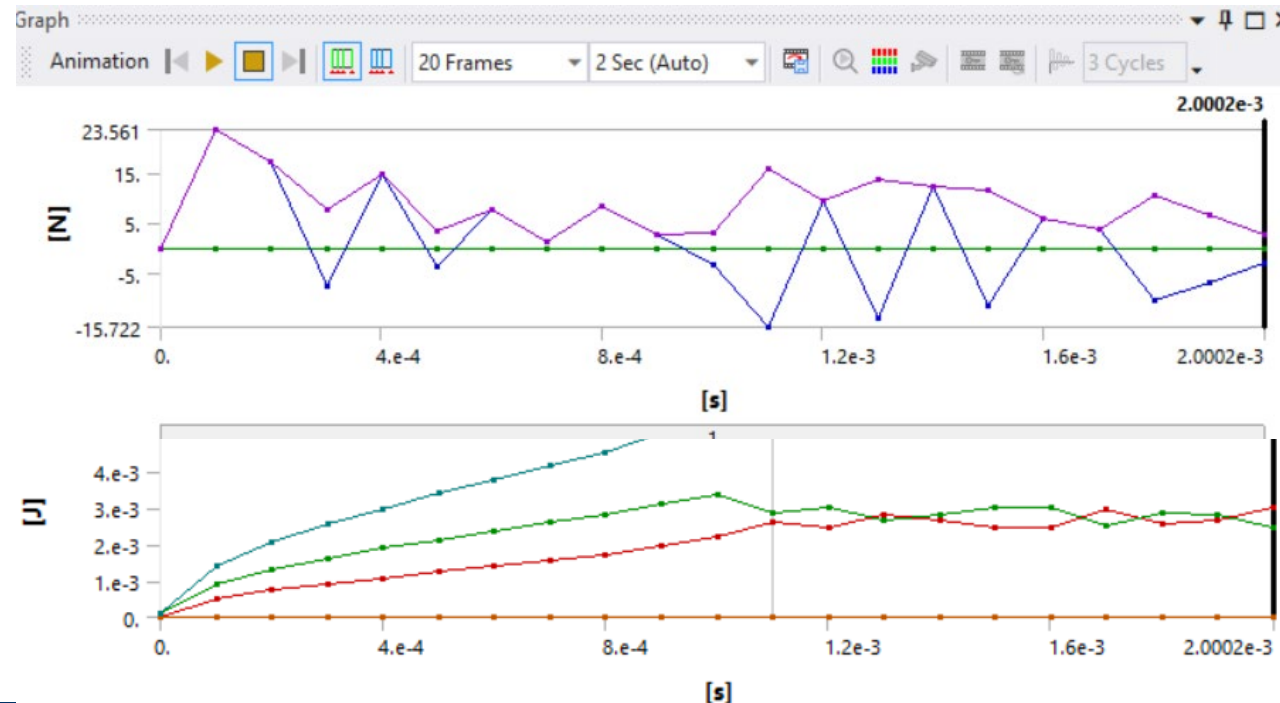
Aplicar un desplazamiento de 1[mm] en Z como una viga en voladizo en 1[us] y mantener hasta 2[us]. ¿Qué fuerza necesitamos? ¿Qué tensión tendremos? ¿Qué pasa tras el primer us?

### Viga en voladizo

$$z_L = \frac{FL^3}{3EI} = \frac{4FL^3}{Ebh^3}$$

$$\sigma_{\max} = M_{\max} \frac{h}{2I} = FL \frac{6}{bh^2}$$

	A	B	C	D
1	input	E	Pa	2.00E+11
2	input	Lx	m	0.3
3	input	Ly	m	0.02
4	input	Lz	m	0.001
5	output	F-z1mm	N	1.48E-01
6	output	s-z1mm	Pa	1.33E+07
7	output	E-z1mm	J	9.88E+05



- **Resumen.**

- Presentación asignatura.
- Necesidad de espacio para poder frenar con poca fuerza y aceleración
- Caso de tracción en explícito e implícito
- Problema de tamaño de malla y necesidad de usar SHELL.

S02t.- Plasticidad e impacto.

Mejora 2122....

## • Repaso última sesión

Importancia de conversión de unidades.

Propiedades de material: densidad, módulo de Young, curva plástica, Poisson, amortiguamiento...

Impactos suelen ocurrir en pocos milisegundos.

Si el material es más duro da más aceleración y si es más blando puede chocar con la pared, El factor de seguridad es tener más espacio.

La energía cinética se transforma en energía de deformación.

Se suele usar el mallado de superficie tipo SHELL para que pueda acabar el cálculo en un tiempo razonable.

Importancia de tamaño de malla y time step. Para acero de longitud 10[mm] el time step está entorno a 2[us] por lo que para simular 20[ms] se necesitan 10000 pasos de cálculo.

En sesión 1 hay un ejemplo para verificar que hacemos bien la simulación.

## • Ejemplo largas deformaciones

Vamos a estirar 300[mm] una barra de 1x20x300[mm] de acero en 1[ms].  
 (A=20x1=20[mm<sup>2</sup>], L=300[mm])

### Estática pequeña deformación:

deformación:  $\varepsilon = \Delta L / L = 300 / 300 = 1[-] = 100\%$

tensión:  $\sigma = E\varepsilon = 200[\text{GPa}] * 1 = 200[\text{GPa}] = 200000[\text{MPa}]$

fuerza:  $F = \sigma A = 200000 * 20 * 1 = 4000000[\text{N}] = 4000[\text{kN}]$

energía:  $E = F\Delta L / 2 = 4000000 * 0.3 / 2 = 0.6[\text{MJ}]$

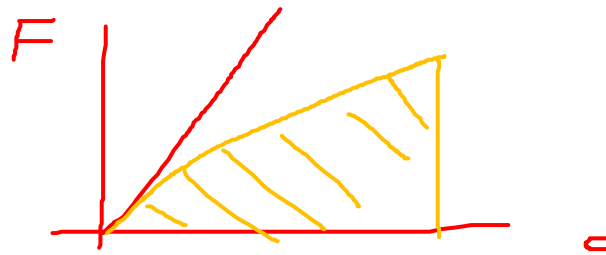
### Estática larga deformación:

deformación:  $\varepsilon = \ln(1 + \Delta L / L) = 300 / 300 = 0.6931[-] = 69.3\%$

tensión:  $\sigma = E\varepsilon = 200[\text{GPa}] * 0.6931 = 139[\text{GPa}] = 139000[\text{MPa}]$

fuerza:  $F = \sigma A * (1 - \nu\varepsilon)^2 = 139000 * 20 * 1 * (1 - 0.3 * 0.6931)^2 = 1740000[\text{N}] = 1740[\text{kN}]$

energía:  $E = F\Delta L / 2 = 1740 * 0.3 / 2 = 0.261[\text{MJ}]$  deberíamos integrar...



# Resultados en ANSYS implícito

- probeta-FreeParts
- Materials
  - Structural Steel
- Coordinate Systems
- Mesh
- Static Structural (A5)
  - Analysis Settings
  - Fixed Support
  - Displacement
  - Displacement 2
  - Displacement 3
  - Displacement 4
- Solution (A6)
  - Solution Information

The deformation is large compared to the model bounding box. Verify boundary conditions or consider turning large deflection on. Refer to Troubleshooting in the Help System for more details.

**A: Static Structural**  
 Equivalent Elastic Strain  
 Type: Equivalent Elastic Strain - Top/Bottom  
 Unit: m/m  
 Time: 1.e-003  
 30/08/2021 16:54 **1 OK**

**A: Static Structural**  
 Equivalent Stress  
 Type: Equivalent (von-Mises) Stress - Top/Bottom  
 Unit: Pa  
 Time: 1.e-003  
 30/08/2021 16:53

**2e+11 Max**  
**2e+11 Min** **2e11 OK**

**A: Static Structural**  
 Equivalent Elastic Strain  
 Type: Equivalent Elastic Strain - Top/Bottom  
 Unit: m/m  
 Time: 1.e-003  
 30/08/2021 17:01

**0.69321 Max** **0.6931 OK**

**A: Static Structural**  
 Equivalent Stress  
 Type: Equivalent (von-Mises) Stress - Top/Bottom  
 Unit: Pa  
 Time: 1.e-003  
 30/08/2021 17:02

**1.3864e+11 Max**  
**1.3864e+11 Min** **1.39e11 OK**

Details of "Analysis Settings"

**Step Controls**

Number Of Steps	1.
Current Step Number	1.
Step End Time	1.e-003 s
Auto Time Stepping	Program Controlled

**Solver Controls**

Solver Type	Program Controlled
Weak Springs	Off
Solver Pivot Checking	Program Controlled
Large Deflection	Off → δN
Inertia Relief	Off
Quasi-Static Solution	Off

**Maximum Value Over Time**

<input type="checkbox"/> X Axis	-4.e-006 N
<input type="checkbox"/> Y Axis	8.0814e-010 N
<input type="checkbox"/> Z Axis	0. N
<input type="checkbox"/> Total	4.e-006 N

**4e6 OK**

**Maximum Value Over Time**

<input type="checkbox"/> X Axis	-1.8287e+006 N
<input type="checkbox"/> Y Axis	519.49 N
<input type="checkbox"/> Z Axis	0. N
<input type="checkbox"/> Total	1.8287e+006 N

**1.74e6 nOK**

**Results**

<input type="checkbox"/> Minimum	10000 J
<input type="checkbox"/> Maximum	10000 J
<input type="checkbox"/> Total	6.e+005 J

**6e5 OK**

**Results**

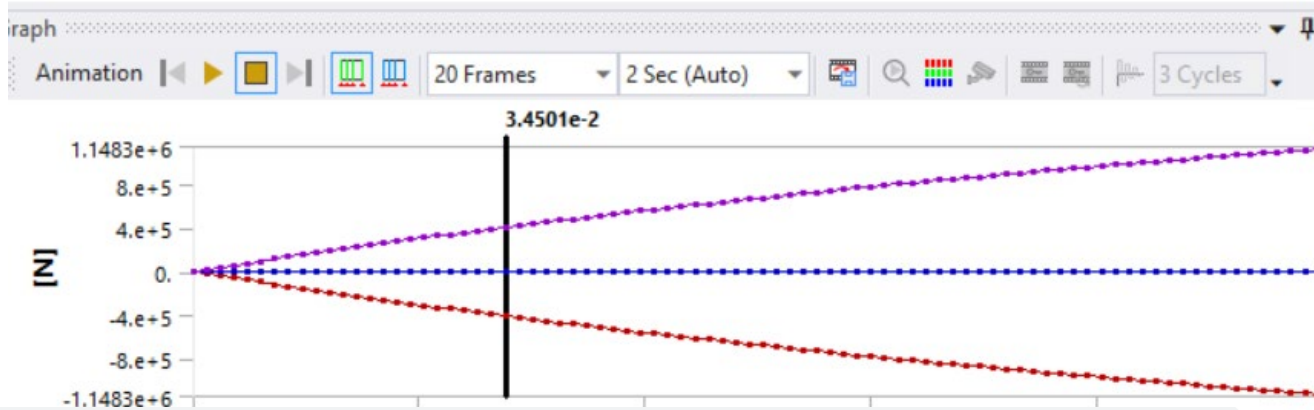
<input type="checkbox"/> Minimum	6338.6 J
<input type="checkbox"/> Maximum	6338.6 J
<input type="checkbox"/> Total	3.8031e+005 J

**2.61e5 nOK**

[https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v212/en/exd\\_ag/ds\\_plasticity.html?q=true%20stress](https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v212/en/exd_ag/ds_plasticity.html?q=true%20stress)

# • Resultados en ANSYS Explícito cargando en 0.001[s]

Encontramos el error de energy error too large.



forum.ansys.com/discussion/3363/turning-simulation#comment-4141f19e-9a0d-44f8-9824-a98b01036366  
 stops the calculation in Explicit Dynamics and let user examine the results before running the calculation further.

Usually, you would need to pay attention to the energy balance if there is a spike in the energy history curve under Solution -> Solution Information -> Energy Conservation.

If the energy error gradually increases, usually the results are OK. If the energy error goes up very quickly in very few cycles, you need to examine the results and the model setup very carefully since it usually indicates the problems with the model setup or the solutions. There could also be problems in the result due to eroded elements, element sizes not small enough and other similar symptoms.

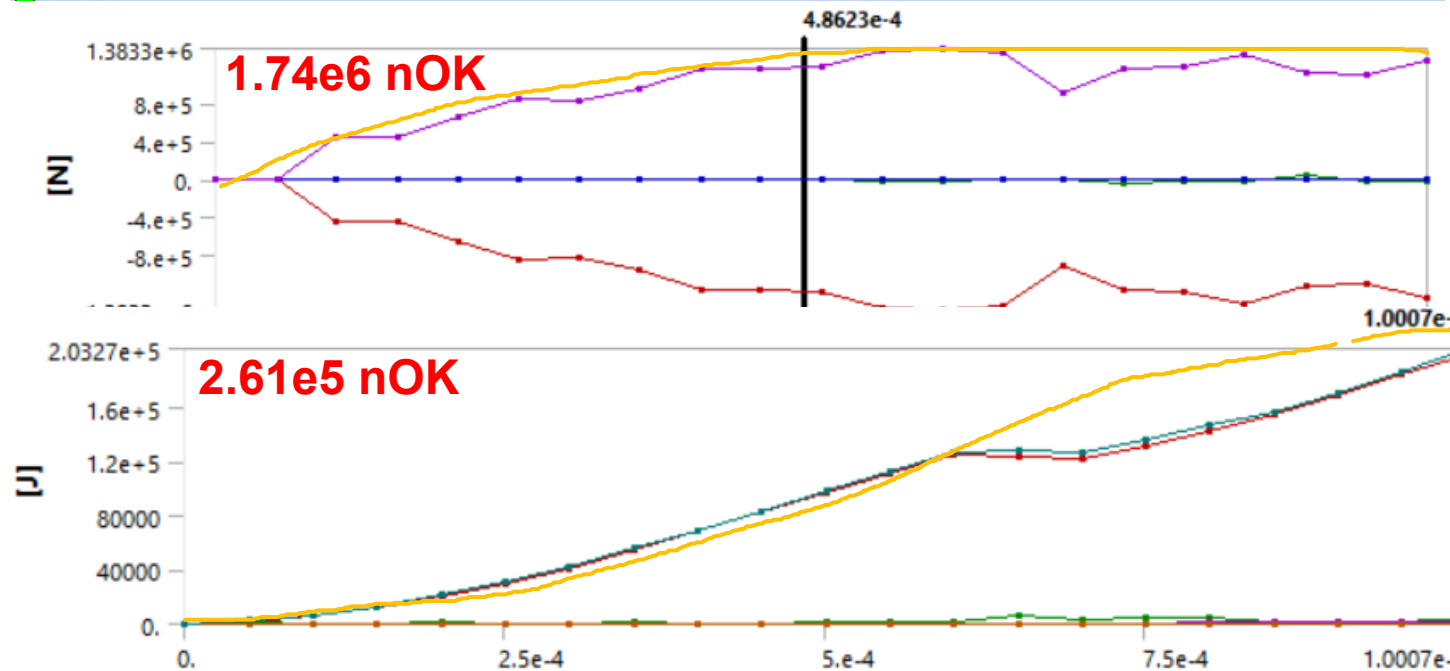
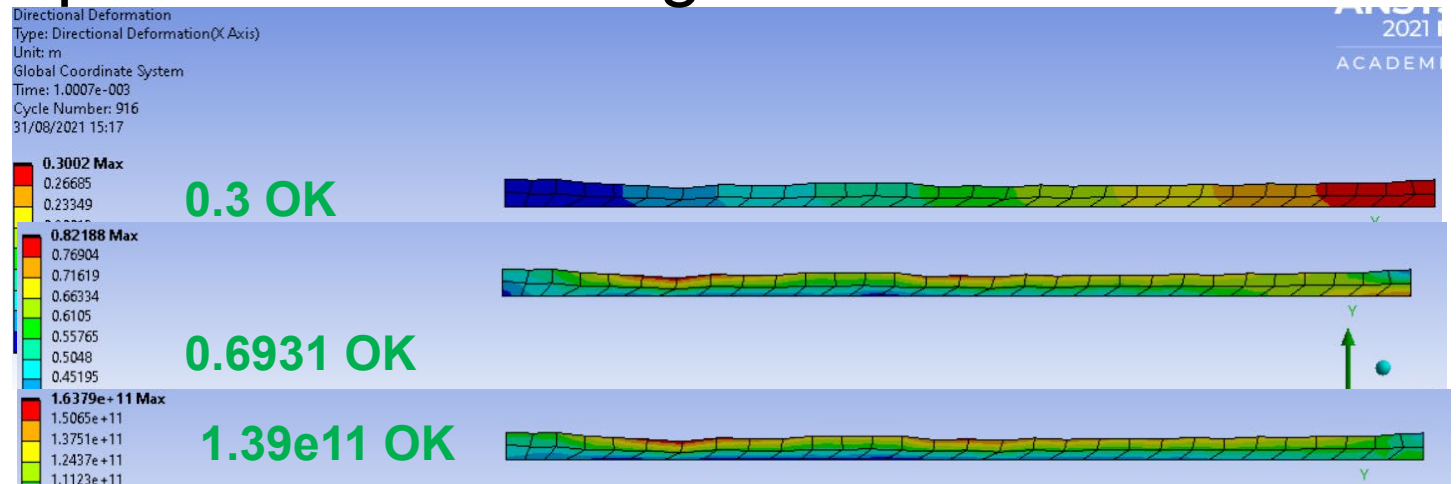
Now, If you think the energy balance looks ok and just want to get the problem to run you can Disable the energy balance check by setting the Reference Energy Cycle to the same number as Maximum Number of Cycles (default 1E+7) in Step Controls. This disables the energy error check in Explicit Dynamics and Autodyn Component systems. The program will still calculate the energy and the energy error but it won't interrupt the solving process when the energy error exceeds the specified value.

Details of "Analysis Settings"	
<b>Analysis Settings Preference</b>	
Type	Program Controlled
<b>Step Controls</b>	
Number Of Steps	1
Current Step Number	1
Load Step Type	Explicit Time Integrati
End Time	0.3
Resume From Cycle	0
Maximum Number of Cycles	1e+07
<b>Maximum Energy Error</b>	<b>1.e+007</b>
Reference Energy Cycle	0
Initial Time Step	Program Controlled
Minimum Time Step	Program Controlled
Maximum Time Step	Program Controlled



# • Solución sin comprobación de energías

En la solución explícita con tanta deformación no se consigue un movimiento uniforme con tensiones muy similares como en el implícito.



## • Plasticidad

Vamos a estirar 3[mm] una barra de 1x20x300[mm] de acero en 1[ms].  
( $A=20 \times 1=20[\text{mm}^2]$ ,  $L=300[\text{mm}]$ ) pero que plastifica a los 200[MPa]

### Estática pequeña deformación:

deformación:  $\varepsilon = \Delta L / L = 3 / 300 = 0.01[-] = 1\%$

tensión:  $\sigma = E\varepsilon = 200[\text{GPa}] * 0.01 = 2[\text{GPa}] = 2000[\text{MPa}]$  pero como plastifica se quedará en 200[MPa].

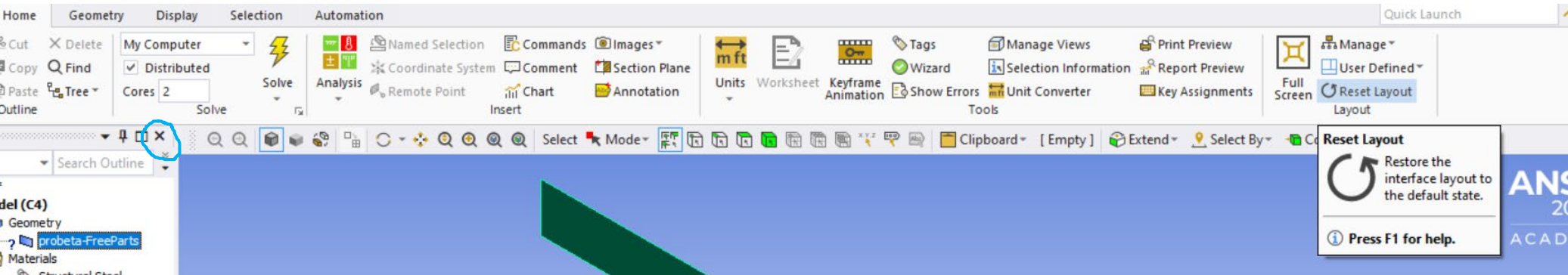
fuerza:  $F = \sigma A = 200 * 20 * 1 = 4000[\text{N}] = 4[\text{kN}]$

energía:  $E = F\Delta L = 4000 * 0.003 = 12[\text{J}]$  ahora es una zona plana.

$$12 - \frac{12}{20} = 11.4[\text{J}]$$



En Ansys si cierras por accidente una ventana hacer Reset Layout



# • Cambio material

Outline of Schematic C2: Engineering Data

	A	B	C	D	E
1	Contents of Engineering Data		Source	Description	
2	Material				
3	Structural Steel				Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1
*	Click here to add a new material				

Properties of Outline Row 3: Structural Steel

	A	B	C	D	E
1	Property		Value	Unit	
2	Material Field Variables				
3	Density	7850		kg m <sup>-3</sup>	
4	Isotropic Secant Coefficient of Thermal Expansion				
6	Isotropic Elasticity				
12	Bilinear Isotropic Hardening				
13	Yield Strength	2E+08		Pa	
14	Tangent Modulus	0		Pa	
15	Strain-Life Parameters				
23	S-N Curve				
27	Tensile Yield Strength	2.5E+08		Pa	
28	Compressive Yield Strength	2.5E+08		Pa	
29	Tensile Ultimate Strength	4.6E+08		Pa	
30	Compressive Ultimate Strength	0		Pa	

Table of Properties Row 12: Bilinear Isotropic Hardening

	A	B	C
1	Temperature (C)	Yield Strength (Pa)	Tangent Modulus (Pa)
2		2E+08	0
*			

Chart of Properties Row 12: Bilinear Isotropic Hardening

C: Static-plasticity-200MPa  
 Equivalent Stress  
 Type: Equivalent (von-Mises) Stress - Top/Bottom  
 Unit: Pa  
 Time: 1.e-003  
 01/09/2021 13:36

2e8 OK

<input type="checkbox"/> Display Time	End Time
<b>Results</b>	<b>4e3 OK</b>
<b>Maximum Value Over Time</b>	
<input type="checkbox"/> X Axis	-4000. N
<input type="checkbox"/> Y Axis	-1.7418e-008 N
<input type="checkbox"/> Z Axis	-1.0303e-014 N
<input type="checkbox"/> Total	4001.5 N
<b>Minimum Value Over Time</b>	
<input type="checkbox"/> X Axis	-4001.5 N

Type	Strain Energy
By	Time
<input type="checkbox"/> Display Time	Last
Calculate Time History	Yes
Identifier	
Suppressed	No
<b>Results</b>	
<input type="checkbox"/> Minimum	0.19 J
<input type="checkbox"/> Maximum	0.19 J
<input type="checkbox"/> Total	<b>11.4 OK</b>

# • Repetimos en explícito

## D: Explicit-plasticity-200MPa

Equivalent Stress

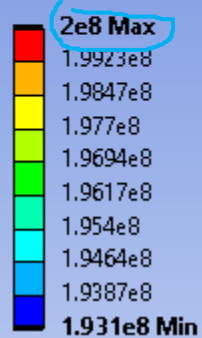
Type: Equivalent (von-Mises) Stress - Top/Bottom

Unit: Pa

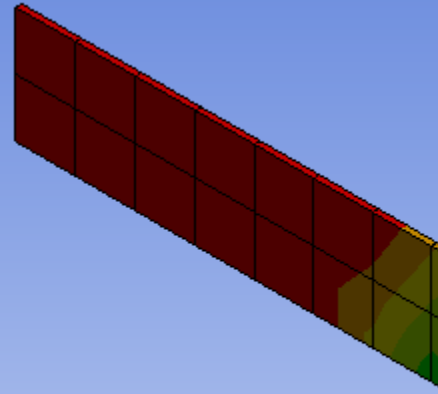
Time: 1.e-003

Cycle Number: 672

01/09/2021 15:26

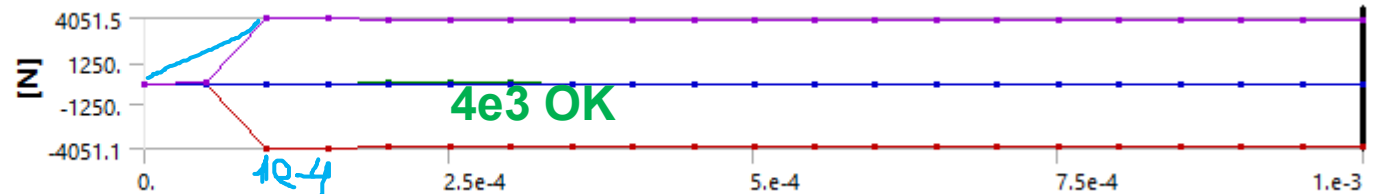


**2e8 OK**



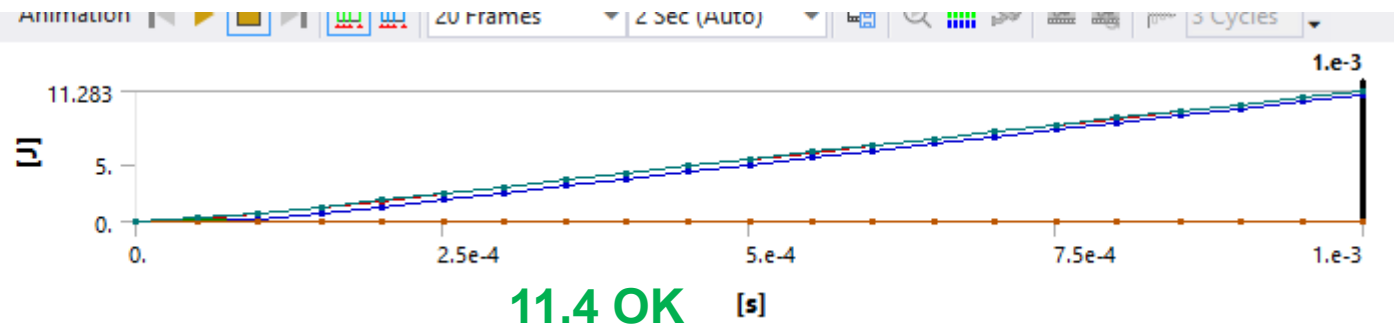
### Maximum Value Over Time

<input type="checkbox"/> X Axis	0. N
<input type="checkbox"/> Y Axis	9.5636 N
<input type="checkbox"/> Z Axis	0. N
<input type="checkbox"/> Total	<b>4051.5 N</b>



### Minimum Value Over Time

<input type="checkbox"/> Internal	11.269 J
<input type="checkbox"/> Kinetic	0.18259 J
<input type="checkbox"/> Plastic Work	10.943 J
<input type="checkbox"/> Hourglass	3.6588e-002 J
<input type="checkbox"/> Contact	0 J
<input type="checkbox"/> Total	<b>11.283 J</b>



- De tracción a impacto masa y velocidad

Para nuestra barra de 1x20x300 su rigidez es  $k=F/X=EA/L=1.33e7[N/m]$ . Si lo consideramos como un muelle y le ponemos una masa de 13.3[kg] en la punta tendremos una frecuencia raíz  $(1.33e7/13.3)=1000[rad/s]$  o  $1.59E+02[hz]$  y por tanto periodo  $6.28E-03[s]$ . Si le colocamos una velocidad de 1[m/s] tendremos la misma energía de 6.67[J] que en el caso elástico en que estirábamos 1[mm]

# • Masa añadida

Project\*  
Model (C4)  
Geometry  
  probeta-FreeParts  
  Point Mass  
Materials  
  Structural Steel  
Coordinate Systems  
Mesh  
Explicit Dynamics (C5)  
  Initial Conditions  
  Pre-Stress (None)  
  Velocity  
  Analysis Settings  
  Displacement  
Solution (C6)  
  Solution Information

**Point Mass**  
02/09/2021 10:15

Point Mass  
Mass Magnitude: 13.3 kg  
Location: 0.155;0;0 m

ANSYS  
2021 R1  
ACADEMIC

Details of "Point Mass"	
<b>Scope</b>	
Scoping Method	Geometry Selection
Applied By	Remote Attachment
Geometry	1 Edge
Coordinate System	Global Coordinate S...
<input type="checkbox"/> X Coordinate	0.155 m
<input checked="" type="checkbox"/> Y Coordinate	0. m
<input type="checkbox"/> Z Coordinate	0. m
Location	Click to Change
<b>Definition</b>	
<input type="checkbox"/> Mass	13.3 kg
<input type="checkbox"/> Mass Moment of Inertia X	1. kg·m <sup>2</sup>
<input type="checkbox"/> Mass Moment of Inertia Y	1. kg·m <sup>2</sup>
<input type="checkbox"/> Mass Moment of Inertia Z	1. kg·m <sup>2</sup>
Suppressed	No
Behavior	Rigid
Pinball Region	All

Geometry	Formula	Geometry	Formula
Solid cylinder or disc, symmetry axis	$I = \frac{1}{2}MR^2$	Hoop about symmetry axis	$I = MR^2$
Solid sphere	$I = \frac{2}{5}MR^2$	Thin spherical shell	$I = \frac{2}{3}MR^2$
Rod about center	$I = \frac{1}{12}ML^2$	Rod about end	$I = \frac{1}{3}ML^2$
Solid cylinder central diameter	$I = \frac{1}{4}MR^2 + \frac{1}{12}ML^2$	Hoop about diameter	$I = \frac{1}{2}MR^2$

# • Velocidad inicial

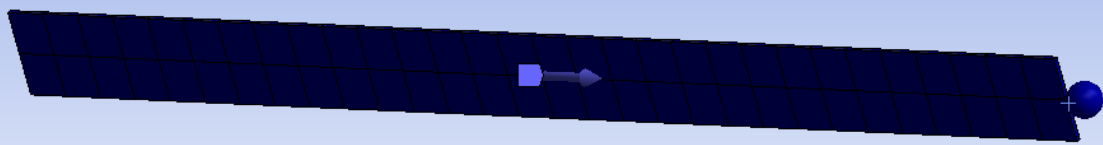
File Home Initial Conditions Display Selection Automation

Duplicate Solve Analysis Coordinate System Remote Point Comment Section Plane Chart Annotation Velocity Angular Velocity Drop Height Graph Tabular Data Views

Outline Name Search Outline

Project\*  
Model (C4)  
Geometry  
probeta-FreeParts  
Point Mass  
Materials  
Structural Steel  
Coordinate Systems  
Global Coordinate System  
Remote Points  
Mesh  
Named Selections  
Explicit Dynamics (C5)  
Initial Conditions  
Pre-Stress (None)  
Velocity  
Analysis Settings  
Displacement  
Solution (C6)  
Solution Information  
Equivalent Stress  
Directional Deformation

C: Explicit-elastic-m 13.3kg-v385m\_s  
Velocity  
02/09/2021 11:41  
Velocity: 1 m/s



0.000 0.025 0.050 0.075 0.100 (m)

details of "Velocity"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Input Type	Velocity
Pre-Stress Environment	None Available
Define By	Vector
<input type="checkbox"/> Total	1. m/s
Direction	Click to Change
Suppressed	No

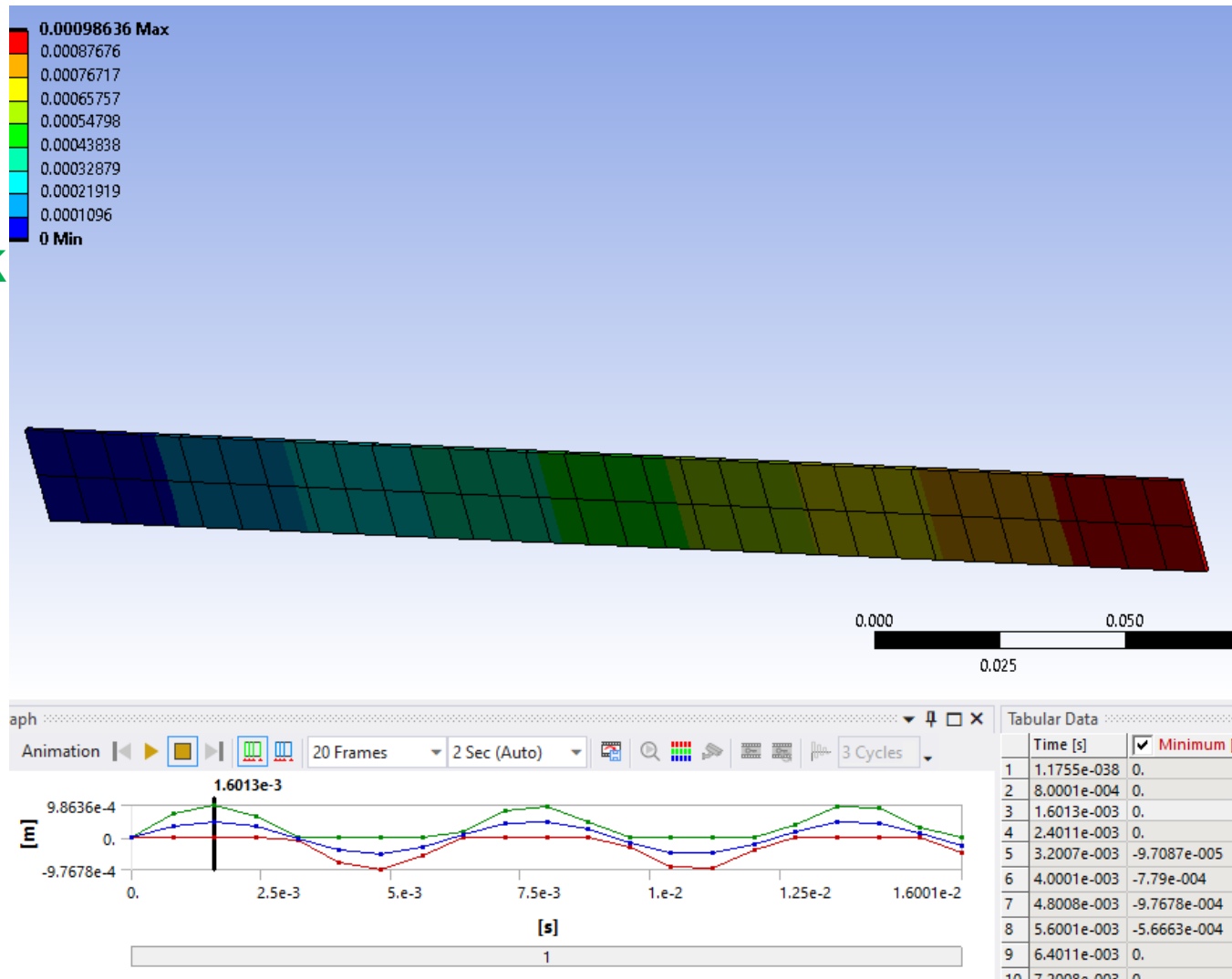
Messages

Text	Association	Timestamp

# • Comprobación

1[mm] OK

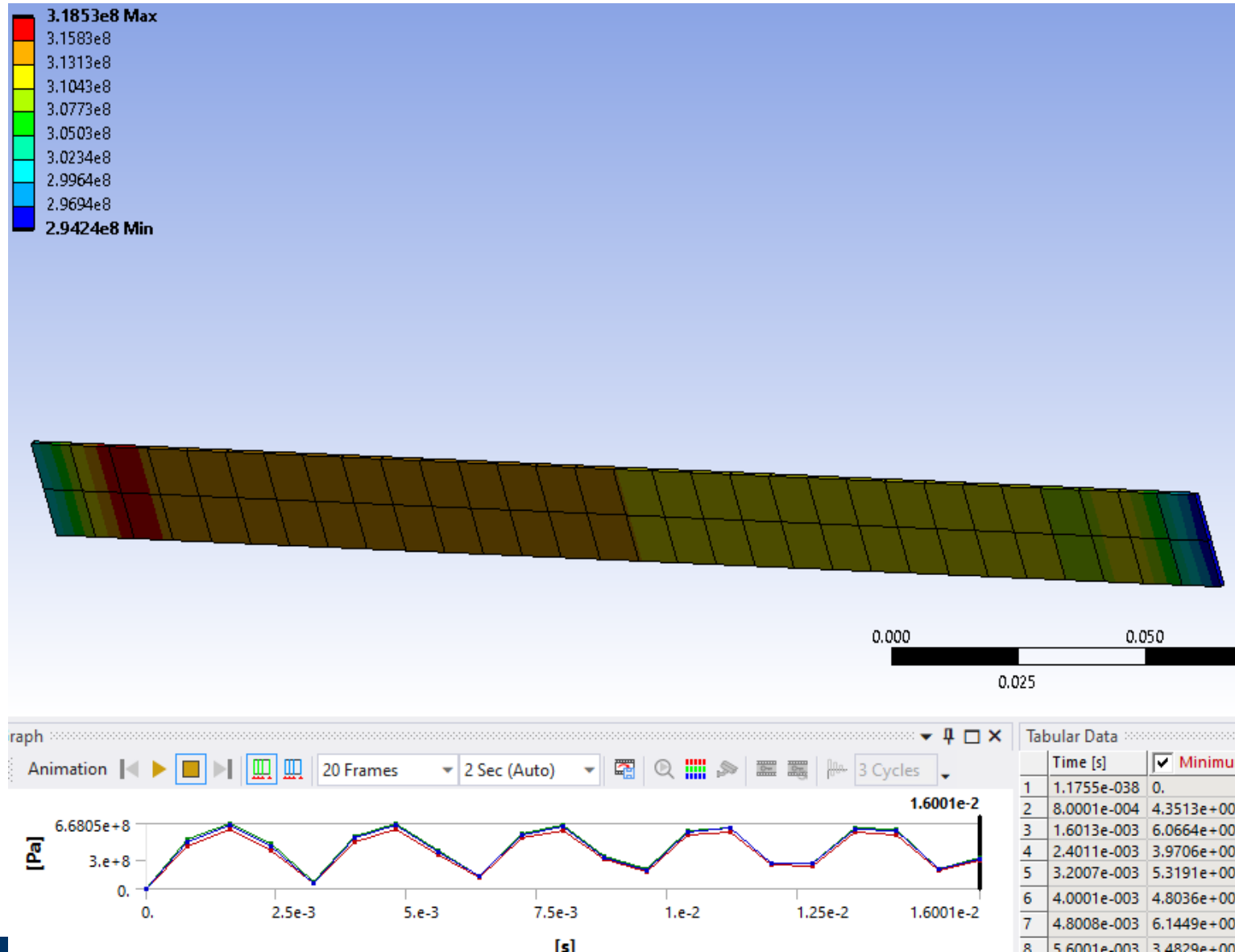
6.28[ms] OK





# • Comprobación

667[MPa] OK



# • Comprobación

6.67[J] OK

Details of "Energy Probe"

Definition	
Type	Energy
Geometry	1 Body
Suppressed	No
Options	
Result Selection	All
<input type="checkbox"/> Display Time	End Time
Results	
Maximum Value Over Time	
<input type="checkbox"/> Internal	6.4981 J
<input type="checkbox"/> Kinetic	6.5232 J
<input type="checkbox"/> Plastic Work	0. J
<input type="checkbox"/> Hourglass	1.8582e-003 J
<input type="checkbox"/> Contact	0. J
<input type="checkbox"/> Total	6.5312 J

Graph

Tabular Data		
	Time [s]	Energy [J]
1	1.1755e-038	0.
2	8.0001e-004	3.4193
3	1.6013e-003	6.4981
4	2.4011e-003	2.7987
5	3.2007e-003	6.7303e-002
6	4.0001e-003	4.0484
7	4.8008e-003	6.3859
8	5.6001e-003	2.1573
9	6.4011e-003	0.26881
10	7.2008e-003	4.6739

Output Controls

Step-aware Output Controls	No
Save Results on	Equally Spac...
Result Number Of Points	200

Graph

Tabular Data		
	Time [s]	Energy [J]
1	1.1755e-038	0.
2	8.0883e-005	4.9065e-002
3	1.6144e-004	0.1774
4	2.4051e-004	0.37952
5	3.2108e-004	0.67023
6	4.0015e-004	1.0164
7	4.8071e-004	1.4258
8	5.6128e-004	1.8915
9	6.4036e-004	2.3776

## • Animated GIF

C: Explicit-elastic-m13.3kg-v385m\_s - Mechanical [ANSYS Mechanical Enterprise]

File Home Result Display Selection Automation

1.0 (True Scale) Scoped Bodies Large Vertex Contours

Geometry Contours Edges Display

Probe Snap Vectors Proportional Element Aligned Grid Aligned

Maximum Minimum Uniform Line Form Solid Form

Y Axis X Axis Z Axis Iso 0 Capped Isosurface

Worksheet Graph Tabular Data Views

Outline

Name Search Outline

- Project\*
  - Model (C4)
    - Geometry
      - probeta-FreeParts
      - Point Mass
    - Materials
      - Structural Steel
    - Coordinate Systems
      - Global Coordinate System
    - Remote Points
    - Mesh
    - Named Selections
    - Explicit Dynamics (C5)
      - Initial Conditions
        - Pre-Stress (None)
        - Velocity
      - Analysis Settings
      - Displacement
    - Solution (C6)
      - Solution Information
      - Normal Stress
      - Directional Deformation
      - Energy Probe

C: Explicit-elastic-m13.3kg-v385m\_s

Normal Stress

Type: Normal Stress(X Axis) - Top/Bottom

Unit: Pa

Global Coordinate System

Time: 1.6001e-002

Cycle Number: 10728

02/09/2021 11:51

-3.0997e8 Max

-3.1061e8

-3.1124e8

-3.1188e8

-3.1251e8

-3.1315e8

-3.1378e8

-3.1441e8

-3.1505e8

-3.1568e8 Min

0.000 0.025 0.050 0.075 0.100(m)

Graph

Animation 20 Frames 2 Sec (Auto) 3 Cycles

Tabular Data

Time [s]	Minimum [Pa]	Maximum [Pa]	Average [Pa]
1	1.1755e-038	0.	0.
2	8.0883e-005	3.8601e+007	8.6611e-007
3	1.6144e-004	7.7274e+007	1.2501e-008
4	2.4051e-004	1.5585e+008	1.6943e-008
5	3.2108e-004	1.9384e+008	2.4156e-008
6	4.0015e-004	2.3106e+008	2.7652e+008
7	4.8071e-004	3.0237e+008	3.2357e-008
8	5.6128e-004	3.3672e+008	3.8329e+008
9	6.4036e-004	3.6984e+008	4.1288e-008
10	7.2093e-004	4.3285e+008	4.6067e-008
11	8.0001e-004	4.5868e+008	5.0368e-008

Details of "Normal Stress"

Scope

Scoping Method: Geometry Selection

Geometry: All Bodies

Position: Top/Bottom

Definition

Type: Normal Stress

Orientation: X Axis

By: Time

Display Time: Last

Coordinate System: Global Coordinate System...

Calculate Time History: Yes

Identifier: Suppressed: No

Integration Point Results

Display Option: Averaged

Average Across Bodies: No

Results

Minimum: -3.1568e+008 Pa

Maximum: 3.0997e+008 Pa

Messages Graph

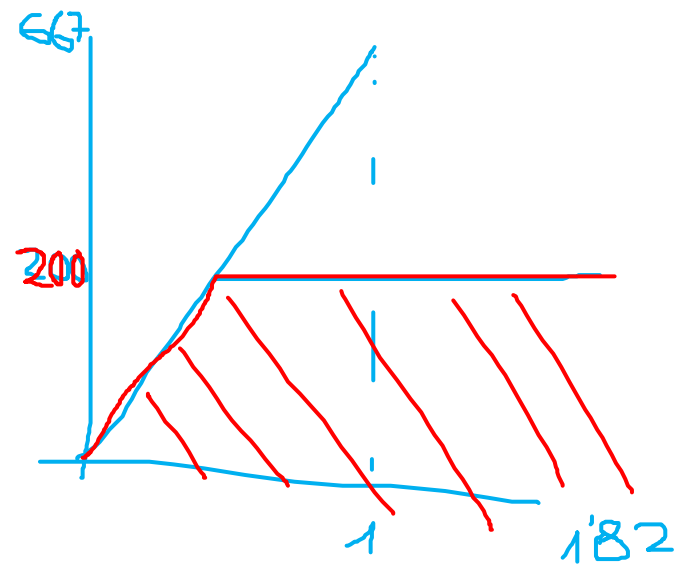
No Messages No Selection Metric (m, kg, N, s, V, A) Degrees rad/s Celsius

11:51 02/09/2021

- Repetir el proceso pero con yield 200 MPa

Ahora esa energía de 6.67[J] tendremos un desplazamiento de 1.82[mm] de la parte elástica más la plástica. (Si aproximamos  $d=E/F$  sale  $6.67/4000=1.67$ [mm]).

Tendremos un pequeño rebote de unos 0.3[mm] de la parte elástica. El tiempo para parar será  $t=v/(F/m)=1/(4000/13.3)=3.33$ [ms].



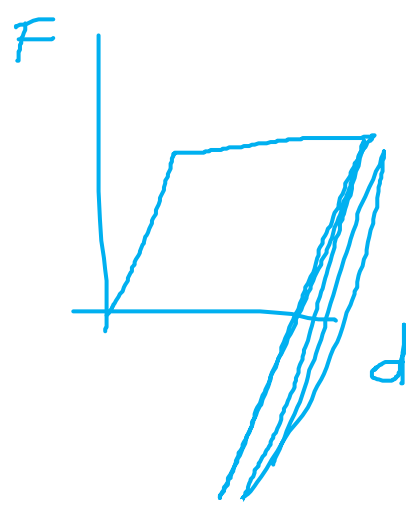
# • Comprobaciones

6.67[J] OK  
3.33[ms] OK

Name Search Outline

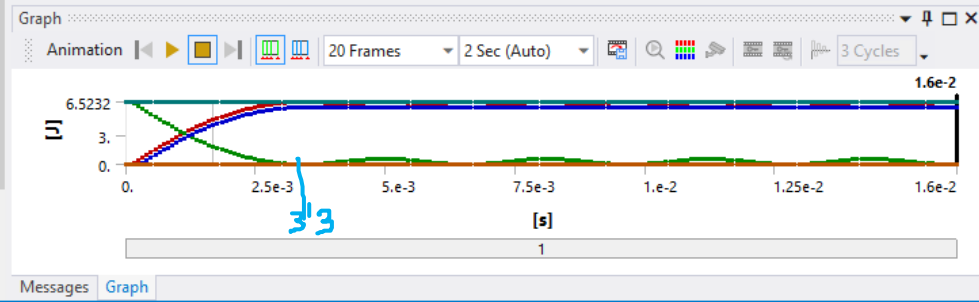
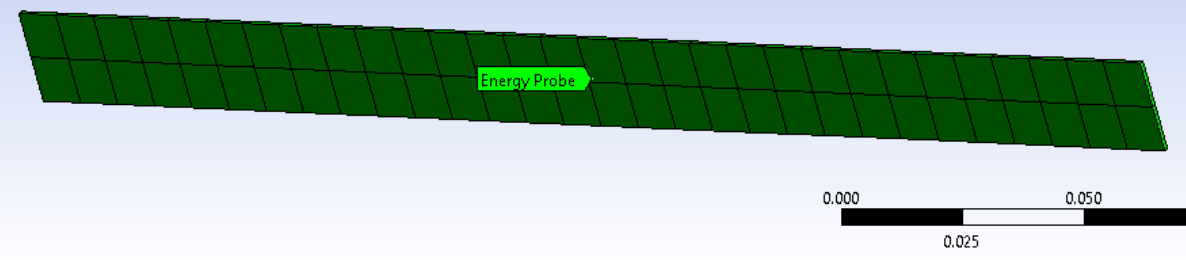
- Project\*
  - Model (F4)
    - Geometry
    - Materials
      - yield200
    - Coordinate Systems
    - Mesh
    - Named Selections
    - Explicit Dynamics (F5)
      - Initial Conditions
      - Analysis Settings
      - Displacement
    - Solution (F6)
      - Solution Information
      - Normal Stress
      - Directional Deformation
      - Energy Probe

F: Explicit-platic-m13.3kg-v1m\_s  
Energy Probe  
02/09/2021 12:11



Details of "Energy Probe"

Definition	
Type	Energy
Geometry	1 Body
Suppressed	No
Options	
Result Selection	All
Display Time	End Time
Results	
Maximum Value Over Time	
Internal	6.5205 J
Kinetic	6.5232 J
Plastic Work	5.9821 J
Hourglass	1.5284e-002 J
Contact	0. J
Total	6.5232 J
Minimum Value Over Time	
Internal	0. J
Kinetic	2.1754e-005 J
Plastic Work	0. J



Tabular Data

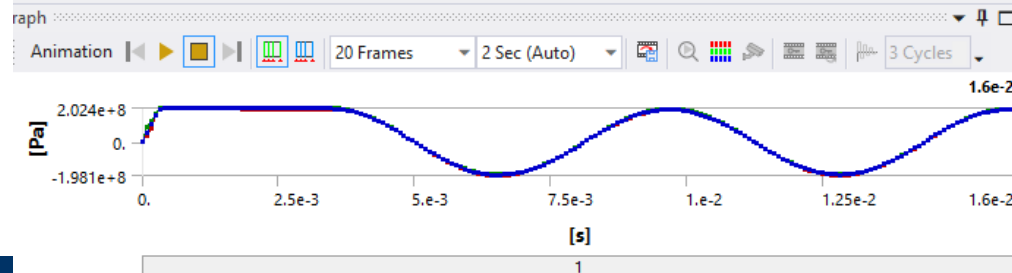
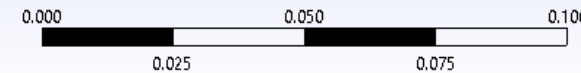
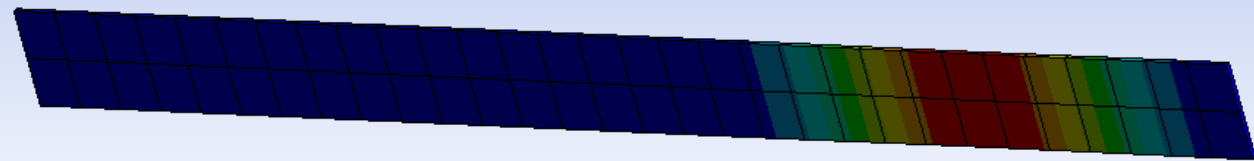
	Time [s]	Energy Pro
1	1.1755e-038	0.
2	8.0883e-005	4.9065e-002
3	1.6144e-004	0.1774
4	2.4051e-004	0.37952
5	3.2108e-004	0.66341
6	4.0015e-004	0.95916
7	4.8072e-004	1.2526
8	5.6129e-004	1.538
9	6.4037e-004	1.8102
10	7.2094e-004	2.0795

# • Comprobaciones

2e8[MPa] OK

F: Explicit-platic-m 13.3kg-v1m\_s  
 Normal Stress  
 Type: Normal Stress(X Axis) - Top/Bottom  
 Unit: Pa  
 Global Coordinate System  
 Time: 1.6e-002  
 Cycle Number: 10727  
 02/09/2021 12:09

1.9677e8 Max  
 1.9627e8  
 1.9577e8  
 1.9527e8  
 1.9478e8  
 1.9428e8  
 1.9378e8  
 1.9328e8  
 1.9279e8  
 1.9229e8 Min

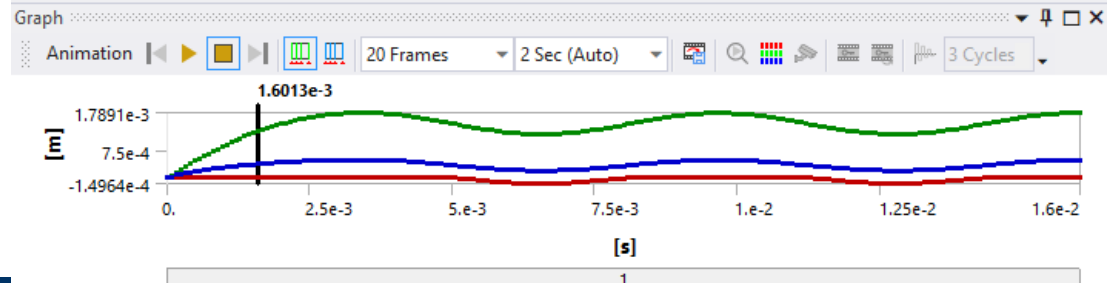
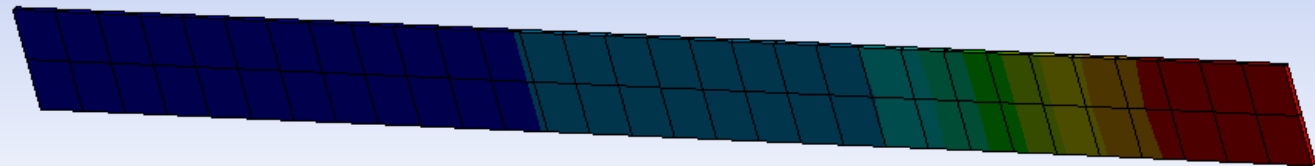
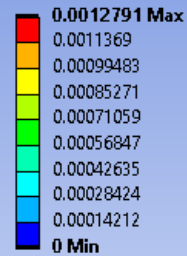


Tabular Data			
	Time [s]	Minimum [Pa]	Maximum [Pa]
1	1.1755e-038	0.	0.
2	8.0883e-005	3.8601e+007	8.6611e+007
3	1.6144e-004	7.7274e+007	1.2501e+008
4	2.4051e-004	1.5585e+008	1.6943e+008
5	3.2108e-004	1.937e+008	2.0243e+008
6	4.0015e-004	1.988e+008	2.0111e+008
7	4.8072e-004	1.9924e+008	2.0057e+008
8	5.6129e-004	1.9945e+008	2.0046e+008
9	6.4037e-004	1.9944e+008	2.0051e+008
10	7.2044e-004	1.9934e+008	2.0051e+008

# • Comprobaciones

1.82 [mm] OK

F: Explicit-platic-m 13.3kg-v1m\_s  
 Directional Deformation  
 Type: Directional Deformation(X Axis)  
 Unit: m  
 Global Coordinate System  
 Time: 1.6012e-003  
 Cycle Number: 1075  
 02/09/2021 12:10



Tabular Data

	Time [s]	Minimum
1	1.1755e-038	0.
2	8.0883e-005	0.
3	1.6144e-004	0.
4	2.4051e-004	0.
5	3.2108e-004	0.
6	4.0015e-004	0.
7	4.8072e-004	0.
8	5.6129e-004	0.
9	6.4037e-004	0.

## • Ejercicio

Realizar la misma simulación con espesor 2 y razonar que pasa.

	A	B	C	D	E
1	input	E	Pa	2.00E+11	2.00E+11
2	input	un	-	3.00E-01	3.00E-01
3	input	Lx	m	0.3	0.3
4	input	Ly	m	0.02	0.02
5	input	Lz	m	0.002	0.001
6	output	F-z1mm	N	1.19E+00	1.48E-01
7	output	s-z1mm	Pa	2.67E+07	1.33E+07
8	output	E-z1mm	J	1.58E+07	9.88E+05
9	output	k-z1mm	N/m	1.19E+03	1.48E+02
10	output	F-x1mm	N	2.67E+04	1.33E+04
11	output	s-x1mm	Pa	6.67E+08	6.67E+08
12	output	E-x1mm	J	1.33E+01	6.67E+00
13	output	k-x1mm	N/m	2.67E+07	1.33E+07
14	input	mp	kg	2.67E+01	1.33E+01
15	output	w	rad/s	1.00E+03	1.00E+03
16	output	w	hz	1.59E+02	1.59E+02
17	output	T	s	6.28E-03	6.28E-03
18	output	v	m/s	1.00E+00	1.00E+00
19	output	E-yield200	J	1.33E+01	6.67E+00
20	output	F-yield200	N	8.00E+03	4.00E+03
21	output	d-yield200	m	3.00E-04	3.00E-04
22	output	dp-yield200	m	1.67E-03	1.67E-03
23	output	dt-yield200	m	1.82E-03	1.82E-03
24	output	t-yield200	s	3.33E-03	3.33E-03
25					
26	output	e-x300mm	-	0.693147	0.693147
27	output	s-x300mm	Pa	1.39E+11	1.39E+11
28	output	F-x300mm	N	2.37E+06	1.18E+06
29	output	E-x300mm	J	3.55E+05	1.78E+05



## • Ejercicio

Hacer el cálculo del peso y velocidad del ensayo documentado de HP en <https://www.youtube.com/watch?v=chTfre8-v5I>

HP Designjet T520 ePrinter Durability Test

110 lb weight

Drop

input	dh	ft	2	1.714	4
input	dw	lb	110	74.18	1
output	dh	m	0.6096	0.522427	1.2192
output	dw	kg	49.89512	33.64745	0.453592
output	E	J	298.2295	172.3557	5.422355
output	v	m/s	3.457493	3.20075	4.889633
output	a-d100mm	m/s <sup>2</sup>	59.77128	51.22399	119.5426
output	a-d13mm	m/s <sup>2</sup>	459.7791	394.0307	919.5582

110 lb weight  
Dropped from 2 ft

Acceleration up to 45 g, 110 lb weight

Impact at 3.2 ft/sec, 74.18 lb

Frequency up to 500 Hz  
Acceleration up to 2.1 g

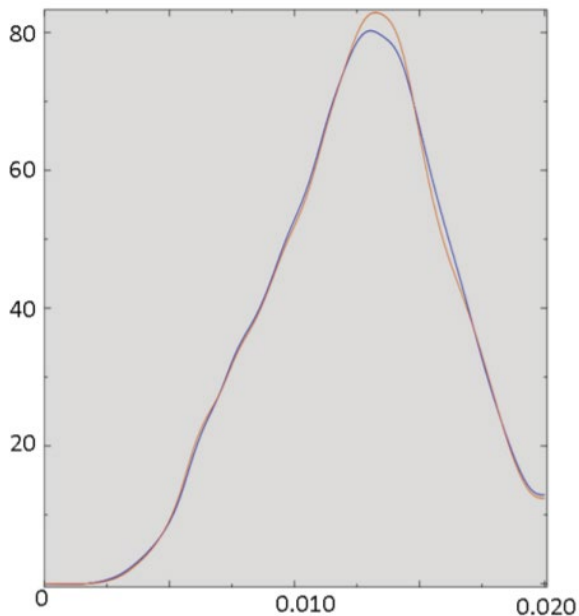
1 lb weight  
Dropped from 4 ft

Masa 1 = 0,453

## • Ejercicio

Buscar información relacionada con los ensayos. Por ejemplo [http://www.ata-e.com/wp-content/uploads/2019/01/HP-Drop-Test-Case-Study\\_2019.pdf](http://www.ata-e.com/wp-content/uploads/2019/01/HP-Drop-Test-Case-Study_2019.pdf)

Otro ejemplo general <https://mpe.researchmfg.com/shipping-carton-dropimpact-test-condition/>



Gross weight, m Kg (lbs.)	Drop height cm (in)
0.0 (0) < m ≤ 9.1 (20)	91 (36)
9.1 (20) < m ≤ 18.2 (40)	76 (30)
18.2 (40) < m ≤ 27.2 (60)	61 (24)
27.2 (60) < m	46 (18)

Simulated print-bar acceleration  
due to drop

- **Resumen.**

- Análisis de largas deformaciones
- Introducción de plasticidad
- Impacto elástico y frecuencias
- Impacto plástico y retorno elástico

## S03t.- Contactos.

Mejora 2122....

- Repaso última sesión

Cálculo con largos desplazamientos.

Cálculo con plasticidad.

Impacto con energía de masa y velocidad elástico y vibraciones y periodos.

Impacto con plasticidad y pequeño retorno elástico

## • Impacto a flexión

Vamos a hacer un impacto a 3 puntos a flexión de una masa y una velocidad. Si el impacto es elástico podemos saber cuanto se va a deformar la viga a flexión.

$$k = 48EI/L^3 = 12Eb^3/L^3.$$

Empezamos con:

$$m = 1 \text{ [kg]}$$

$$v = 0.1 \text{ [m/s]}$$

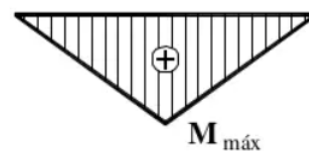
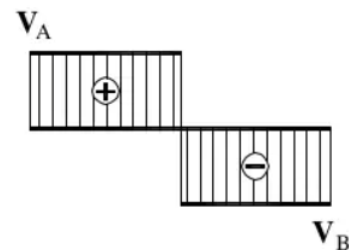
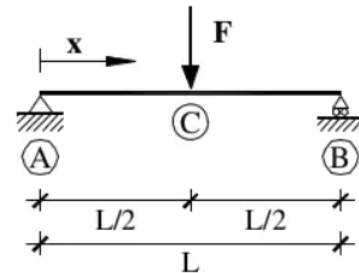
$$L = 0.25 \text{ [m]} \text{ distancia apoyos}$$

$$h = 0.001 \text{ [m]} \text{ espesor } Lz$$

$$b = 0.020 \text{ [m]} \text{ ancho } Ly$$

$$Lx = 0.300 \text{ [m]} \text{ longitud total}$$

### VIGA SIMPLE APOYADA: carga puntual $F$ centrada.



#### Reacciones y solicitaciones

Reacciones:  $R_A = R_B = \frac{F}{2}$

Cortantes:  $V_{AC} = \frac{F}{2}$   $V_{CB} = -\frac{F}{2}$

Flectores:  $M_{AC} = \frac{F}{2}x$   $M_{CB} = \frac{F}{2}(L-x)$

$M_{\text{máx}} = M_C = \frac{FL}{4}$  para  $x = \frac{L}{2}$

#### Deformaciones

Giros:

$$\varphi_A = -\frac{FL^2}{16EI} \quad \varphi_B = \frac{FL^2}{16EI} \quad \varphi_C = 0$$

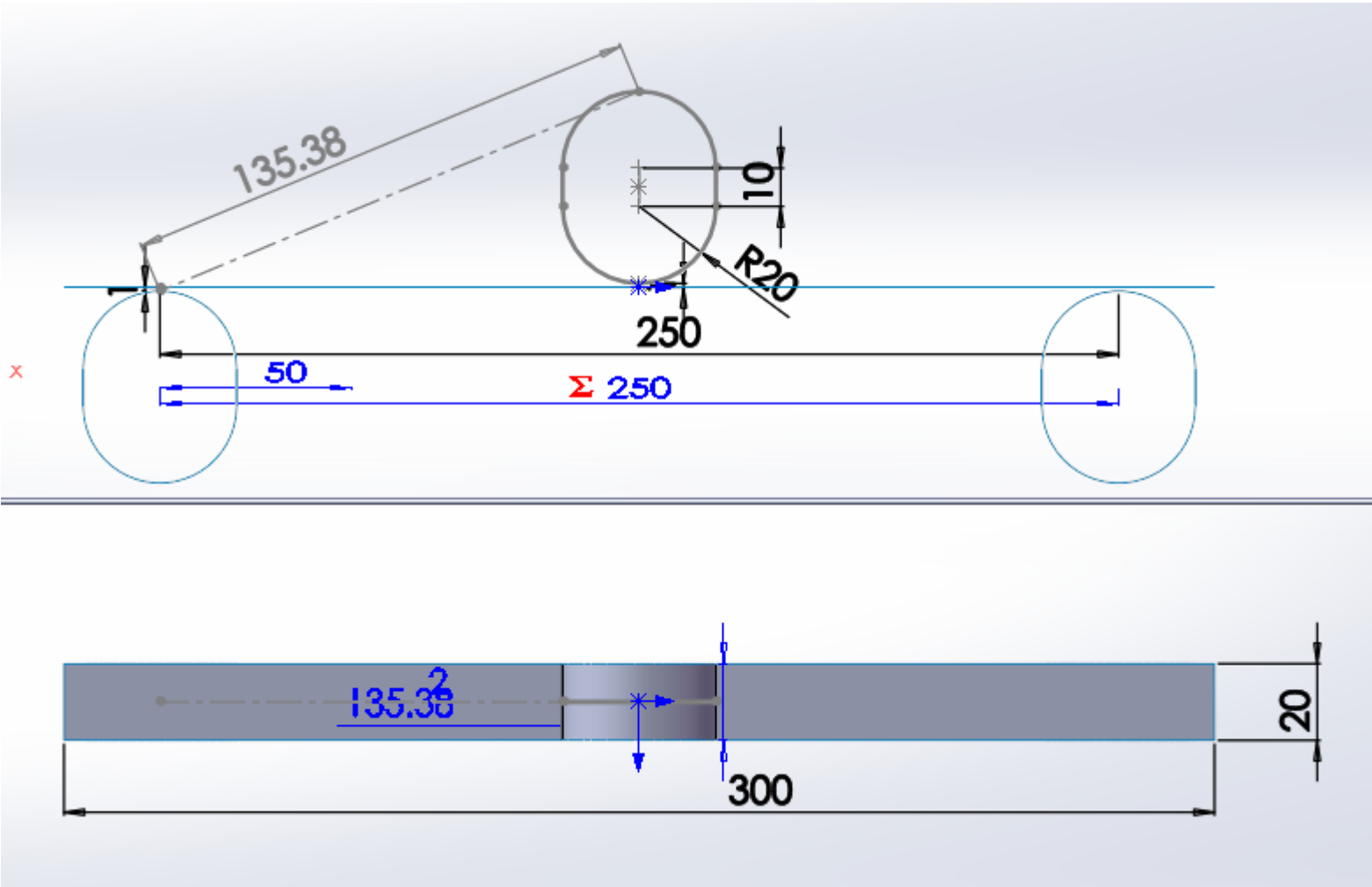
Elástica:

$$y_{AC} = \frac{FL^2 x}{16EI} \left( 1 - \frac{4x^2}{3L^2} \right) \quad y_{CB} = \frac{FL^2(L-x)}{12EI} \left( \frac{3}{4} - \frac{(L-x)^2}{L^2} \right)$$

Flecha máxima:

$$y_{\text{máx}} = y_C = \frac{FL^3}{48EI} \quad \text{para } x = \frac{L}{2}$$

- Geometría impactador a flexión



# • Renombrar

A : Explicit-platic-m13.3kg-v1m\_s - Mechanical [ANSYS Mechanical Enterprise]

File Home Geometry Display Selection Automation

Duplicate Outline Solve Analysis

Named Selection Commands Images Coordinate System Comment Section Plane Remote Point Chart Annotation

Replace Geometry Point Distributed Transform Surface Element Virtual

Geometry Mass Part Coating Orientation Body Virtual

Outline

Name Search Outline

Project\*

Model (A4)

Geometry

Flexion-3P-FreeParts

Materials

Coordinate Systems

Connections

Mesh

Named Selections

Explicit Dynamics (A1)

Initial Conditions

Analysis Settings

Displacement

Solution (A6)

Solution Info

Insert

Update

Generate Mesh

Preview

Freeze Mesh

Hide Body

Hide All Other Bodies

Rename F2

Group

Suppress Bo

Suppress All

Create Name

Create Mate

Create Material Plot

Transform Part

Export...

Update Selected Parts

Clear Generated Data

Rename (F2)

ab Rename the currently selected Outline object.

Press F1 for help.

Details of "Flexion-3P-FreeParts"

Graphics Properties

Definition

Suppressed No

Stiffness Behavior Flexible

Coordinate System Default Coordinate System

Reference Temperature By Environment

Thickness 1.e-003 m

Thickness Mode Manual

Offset Type Middle

Material

Assignment Structural Steel

Messages

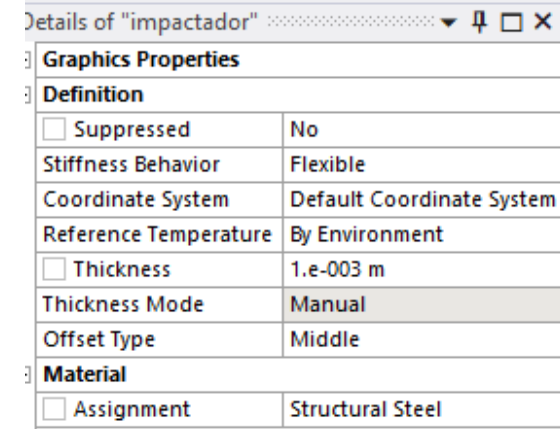
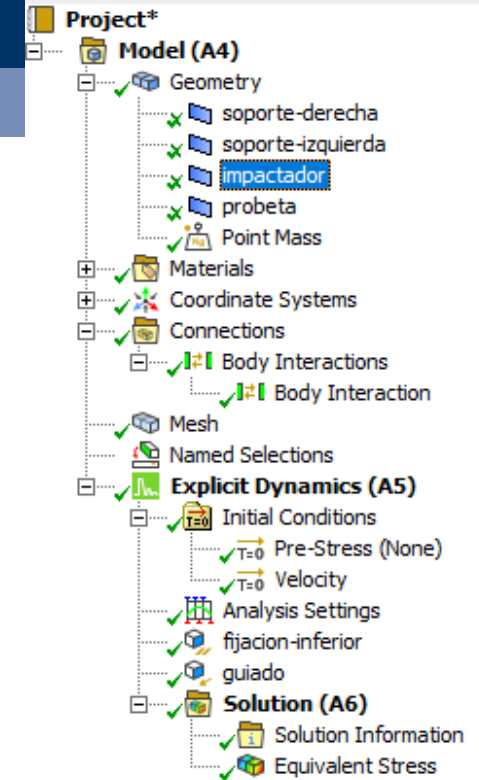
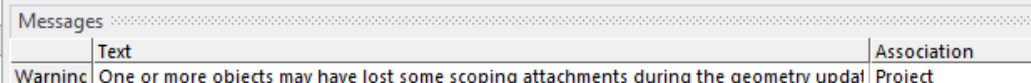
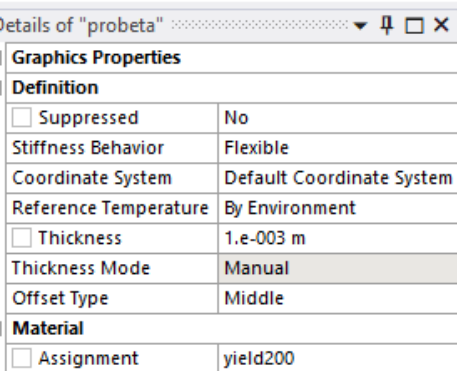
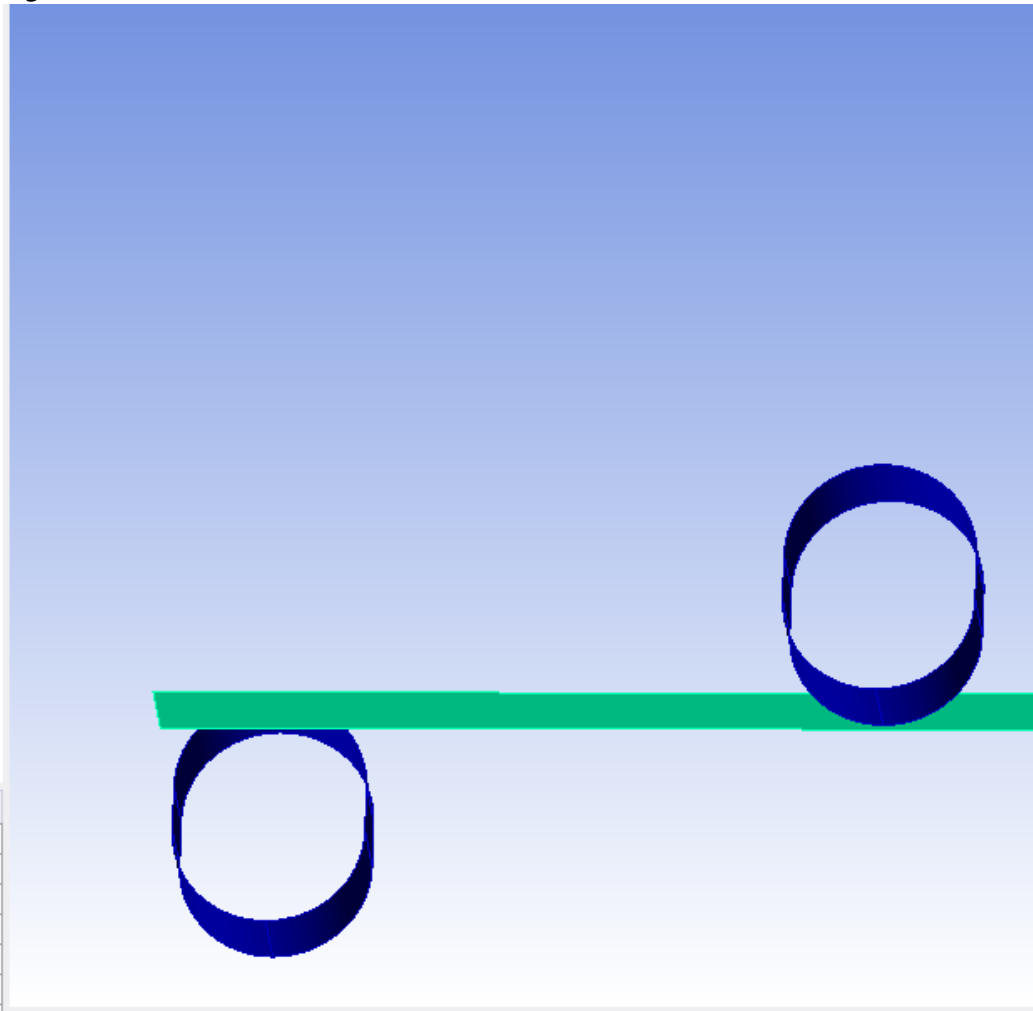
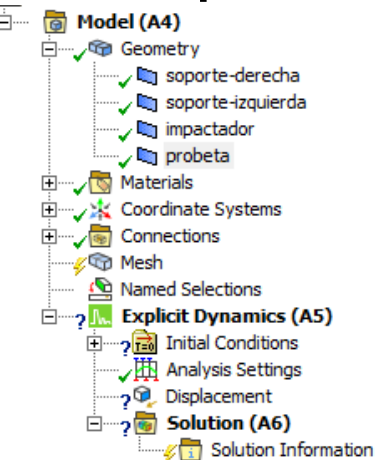
Text	Association	Timestamp
Warning: One or more objects may have lost some scoping attachments during the geometry update	Project	Friday, September 3, 2021 9:22:20 AM

0.000 0.050 0.100(m)

0.025 0.075



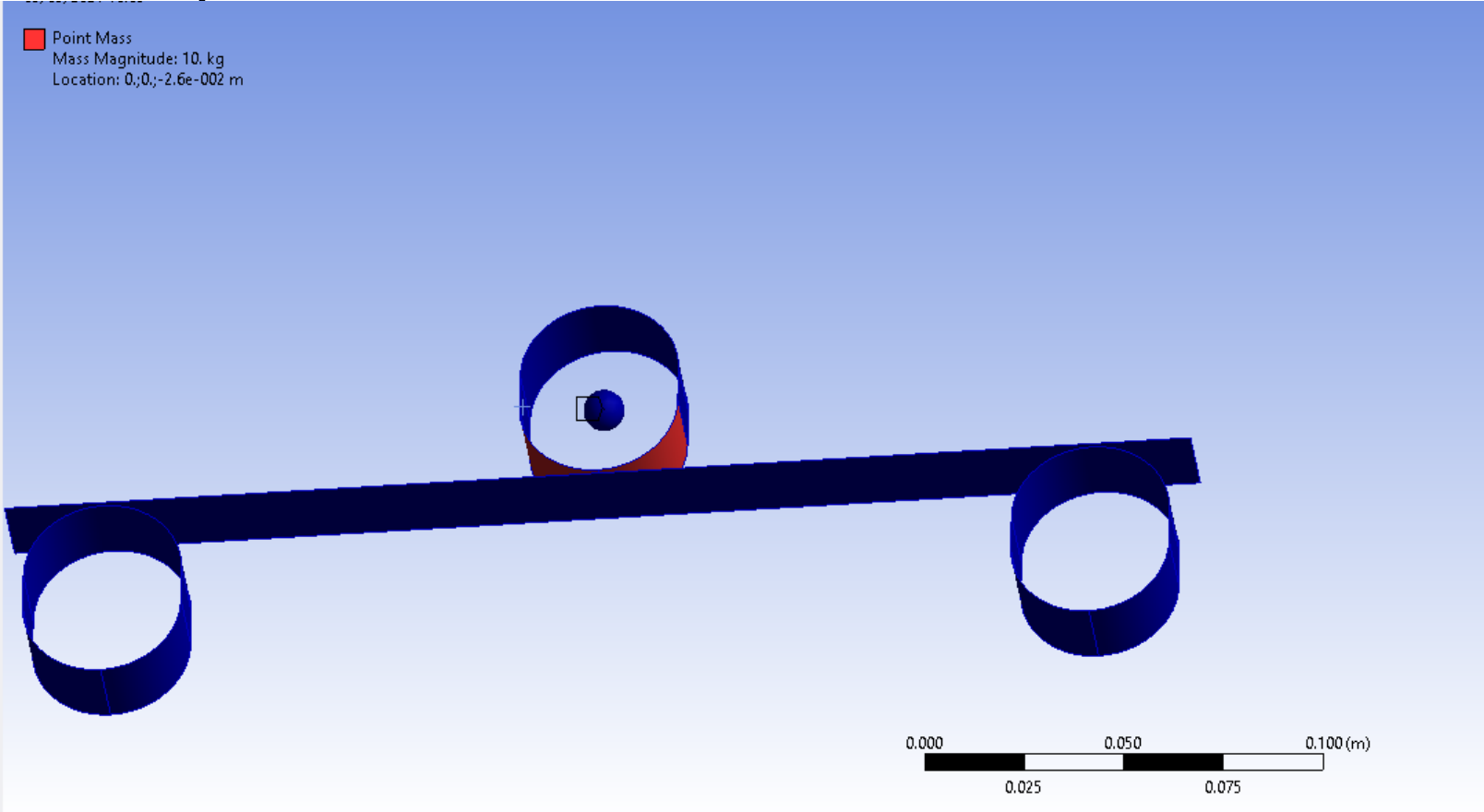
# • Espesores y materiales



# • Aplicar masa impactador

**Model (A4)**

- Geometry
  - soporte-derecha
  - soporte-izquierda
  - impactador
  - probeta
  - Point Mass
- Materials
- Coordinate Systems
- Connections
- Body Interactions
  - Body Interaction
- Mesh
- Named Selections
- Explicit Dynamics (A5)**
  - Initial Conditions
    - Pre-Stress (None)
    - Velocity
  - Analysis Settings
  - fijacion-inferior
  - guiado
- Solution (A6)**
  - Solution Information
  - Equivalent Stress



**Point Mass**  
Mass Magnitude: 10. kg  
Location: 0;0;-2.6e-002 m

0.000      0.050      0.100 (m)  
0.025      0.075

Details of "Point Mass" ⌵ ⌵ ⌵

**Scope**

Scoping Method	Geometry Selection
Applied By	Remote Attachment
Geometry	2 Faces
Coordinate System	Global Coordinate S...
<input type="checkbox"/> X Coordinate	0. m
<input type="checkbox"/> Y Coordinate	0. m
<input type="checkbox"/> Z Coordinate	-2.6e-002 m
Location	Click to Change

**Definition**

<input type="checkbox"/> Mass	10. kg
<input type="checkbox"/> Mass Moment of Inertia X	1. kg·m <sup>2</sup>
<input type="checkbox"/> Mass Moment of Inertia Y	1. kg·m <sup>2</sup>
<input type="checkbox"/> Mass Moment of Inertia Z	1. kg·m <sup>2</sup>
Suppressed	No
Behavior	Rigid
Pinball Region	All

Messages

Text	Association	Timestamp
Warning: One or more objects may have lost some scoping attachments during the geometry update	Project	Friday, September 3, 2021 9:22:20 AM

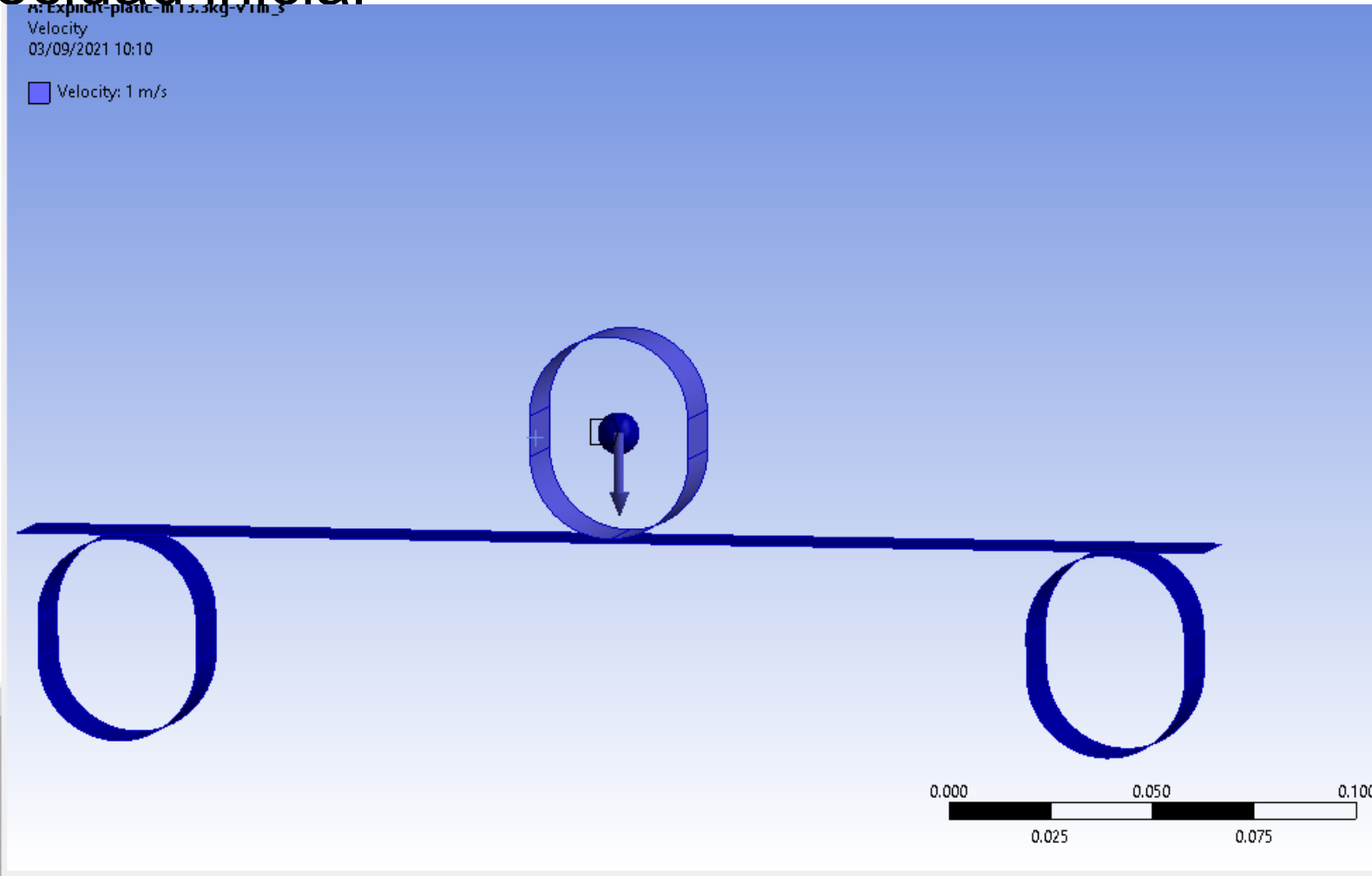
# • Aplicar velocidad inicial

**Project\***

- Model (A4)
  - Geometry
    - soporte-derecha
    - soporte-izquierda
    - impactador
    - probeta
    - Point Mass
  - Materials
  - Coordinate Systems
  - Connections
  - Body Interactions
    - Body Interaction
  - Mesh
  - Named Selections
  - Explicit Dynamics (A5)
    - Initial Conditions
      - Pre-Stress (None)
      - Velocity
    - Analysis Settings
    - fijacion-inferior
    - guiado
  - Solution (A6)
    - Solution Information
    - Equivalent Stress

Velocity  
03/09/2021 10:10

Velocity: 1 m/s



0.000                      0.050                      0.100 (m)

0.025                      0.075

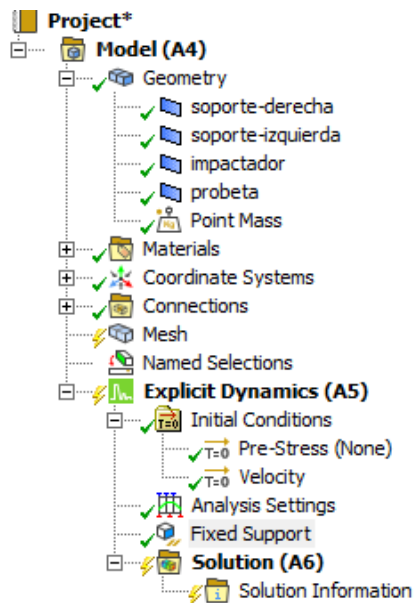
Details of "Velocity" ▾ □ ×

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Input Type	Velocity
Pre-Stress Environment	None Available
Define By	Components
Coordinate System	Global Coordinate System
<input type="checkbox"/> X Component	0. m/s
<input type="checkbox"/> Y Component	0. m/s
<input checked="" type="checkbox"/> Z Component	1. m/s
Suppressed	No

Messages

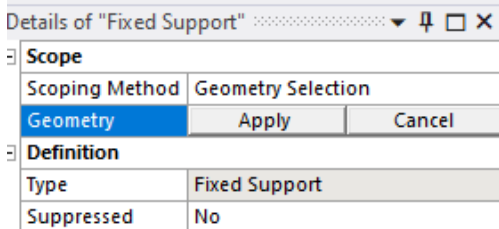
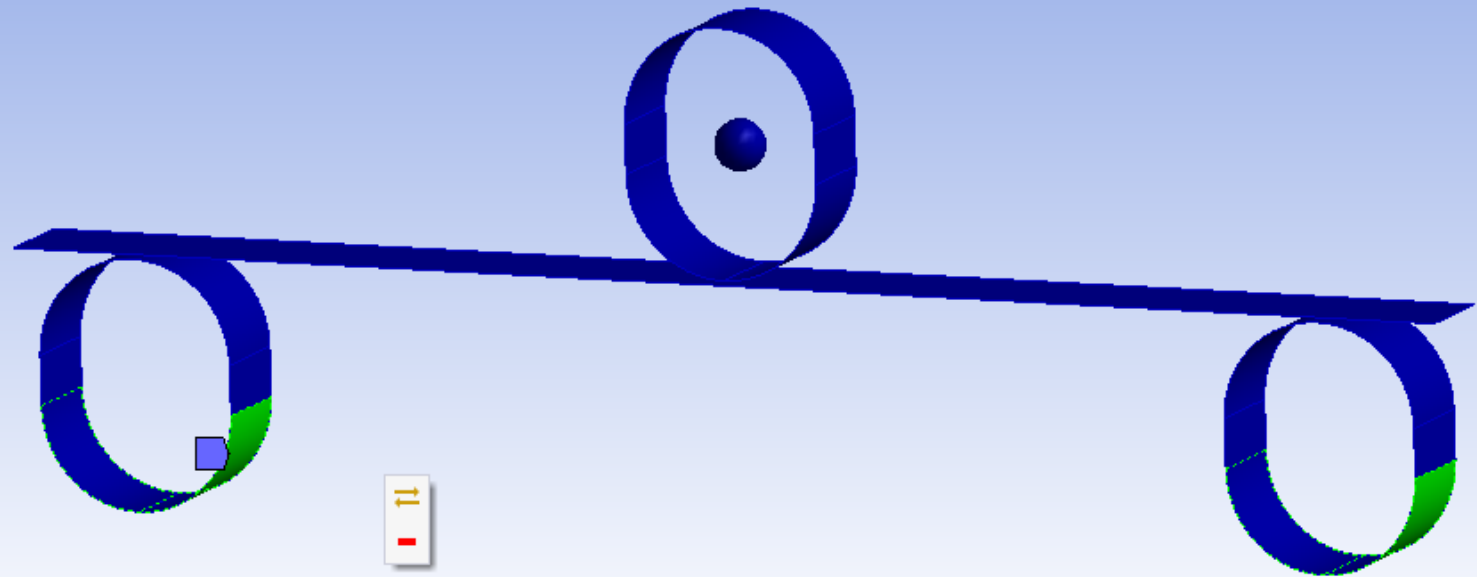
Text	Association	Timestamp
Warning: One or more objects may have lost some scoping attachments during the geometry update	Project	Friday, September 3, 2021 9:22:20

# • Aplicar fijaciones

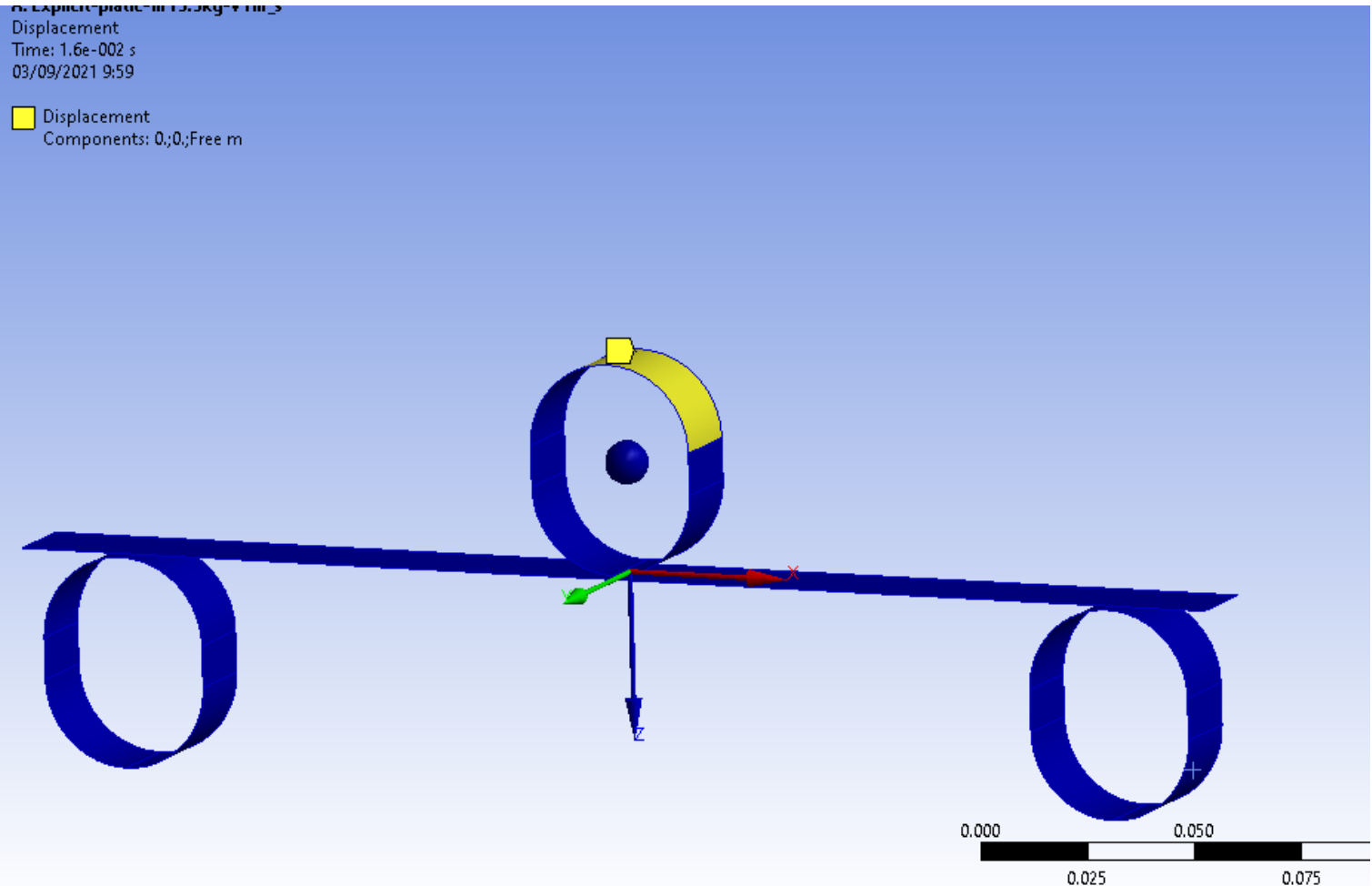
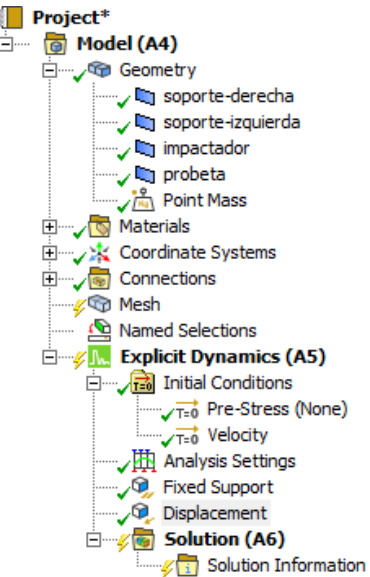


Fixed Support  
Time: 1.6e-002 s  
03/09/2021 9:57

Fixed Support

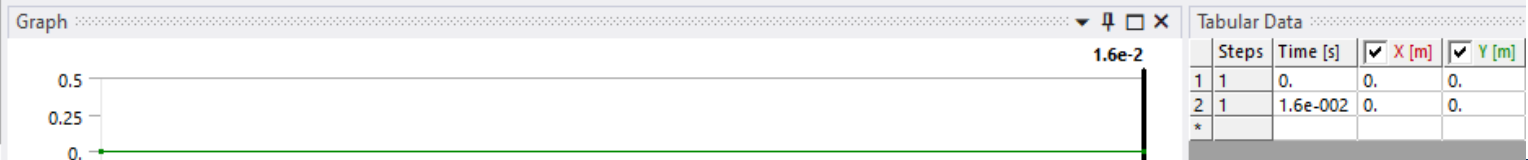


# • Guiar impactador

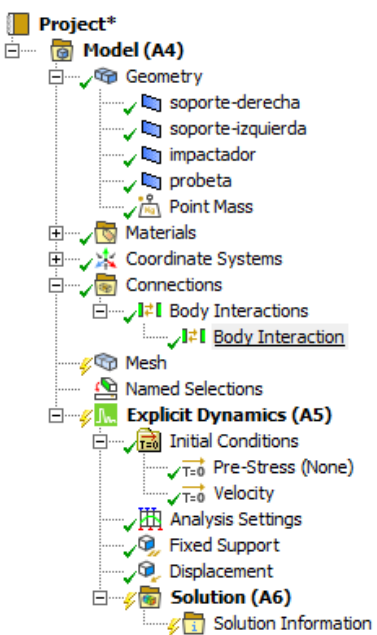


Details of "Displacement"

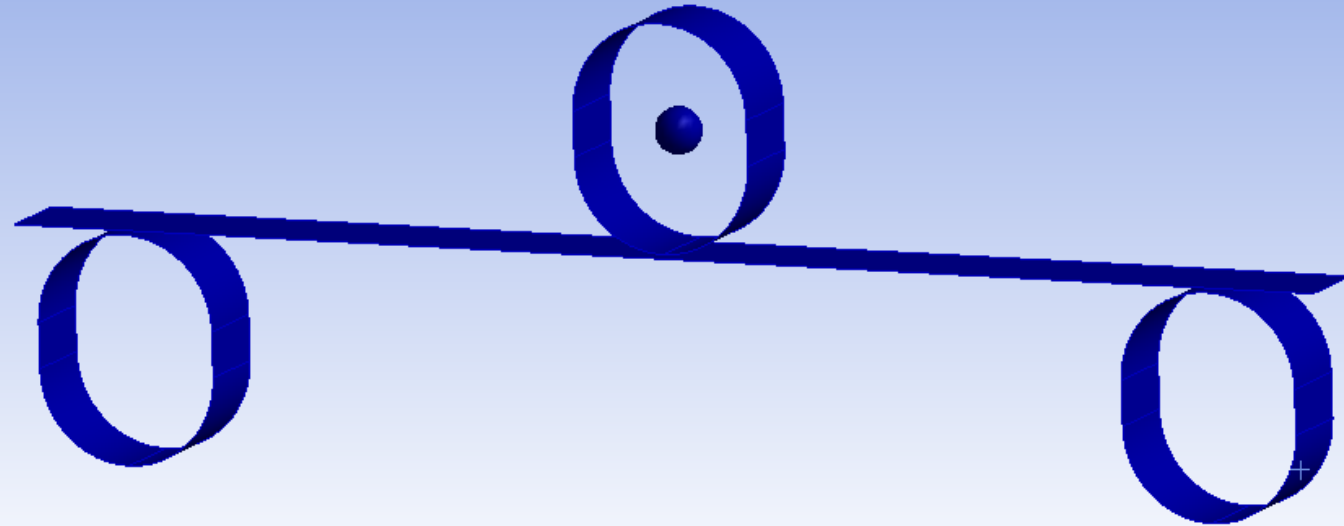
Scope	
Scoping Method	Geometry Selection
Geometry	2 Faces
Definition	
Type	Displacement
Define By	Components
Coordinate System	Global Coordinate System
<input type="checkbox"/> X Component	0. m (ramped)
<input type="checkbox"/> Y Component	0. m (ramped)
Z Component	Free
Suppressed	No






# • Contactos y fricción



Body Interaction  
03/09/2021 10:00




Details of "Body Interaction"   

Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Type	Frictional
Friction Coefficient	0.1
Dynamic Coefficient	0.
Decay Constant	0.
Suppressed	No

Graph   

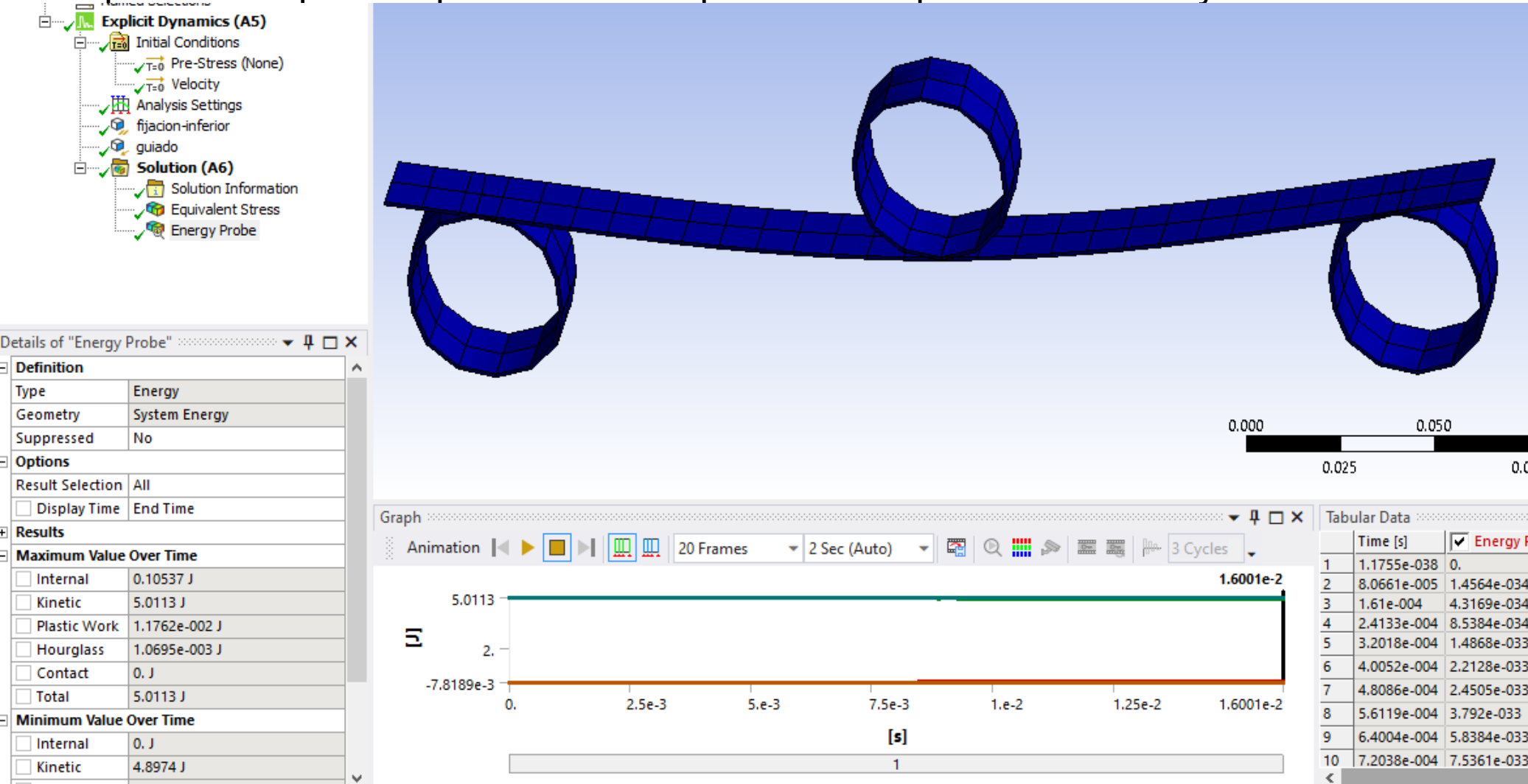
0.000 0.025 0.050 0.075 0.100 (m)

1.6e-2

Tabular Data 

# • Resultados

Se puede comprobar que con este espesor no se para esta masa y velocidad



## • Repetimos con $v=0.1$ [m/s]

Ahora la energía es un 1%, o sea, 0.05[J] y vamos a calcular 1[s] para dar tiempo a acabar. Ahora vemos que necesitamos  $1/1.49e-6 = 671000$  paso de cálculo y ya tarda en poder resolverse un tiempo considerable de uno 8[min] dependiendo del ordenador y número de CPU que utilicemos

```
Cycle: 672174, Time: 1.000E+00s, Time Inc.: 1.488E-06s, Progress: 100.00%, Est. Cl
Cycle: 672175, Time: 1.000E+00s, Time Inc.: 1.488E-06s, Progress: 100.00%, Est. Cl
```

### SIMULATION ELAPSED TIME SUMMARY

```
EXECUTION FROM CYCLE      0 TO    672175
ELAPSED RUN TIME IN SOLVER =    7.52017E+00 Minutes
TOTAL ELAPSED RUN TIME   =    7.92983E+00 Minutes
JOB RAN OVER      2 WORKERS
JOB RAN USING Intel MPI
JOB RAN USING DECOMPOSITION AUTO
```

```
Problem terminated .... wrapup time reached
```

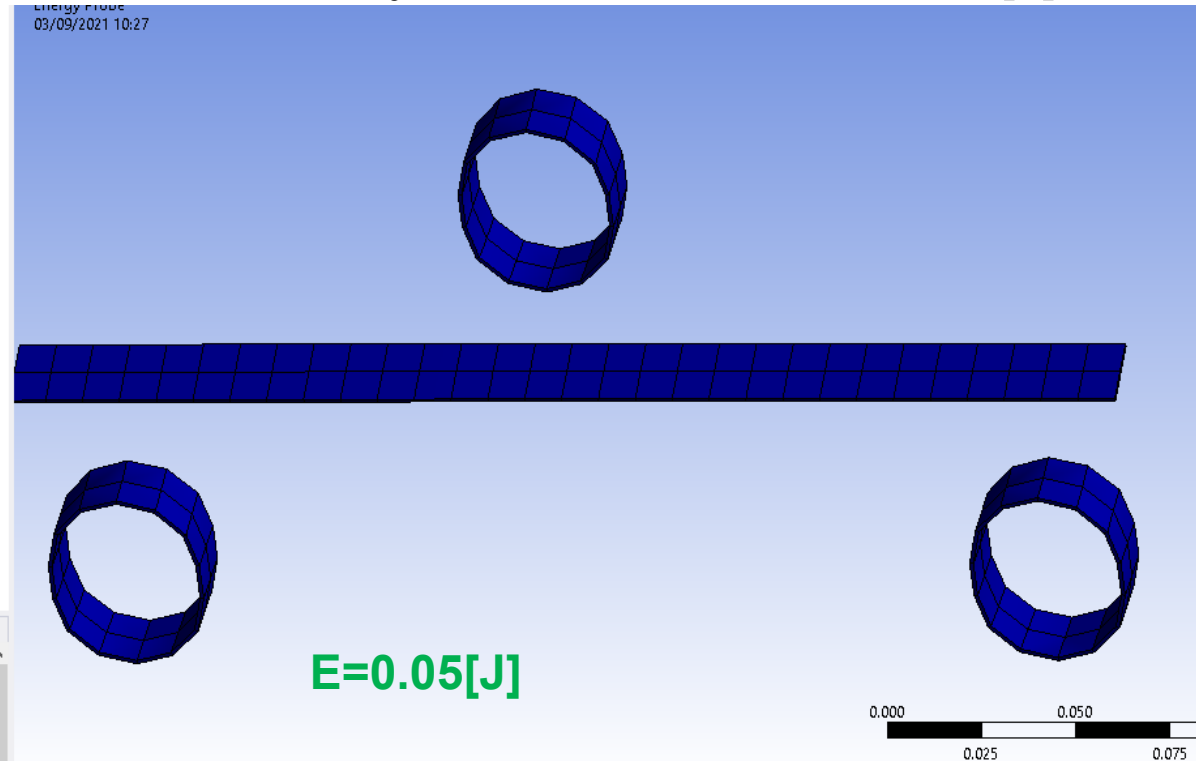
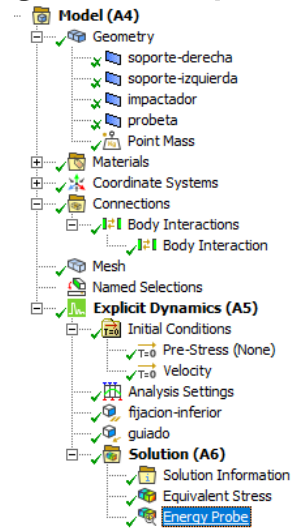
```
*****
```

	A	B	C	D
1	input	E	Pa	2.00E+11
2	input	nu	-	3.00E-01
3	input	Lx	m	0.3
4	input	Ly	m	0.02
5	input	Lz	m	0.001
6	input	D	m	0.25
7	input	m	kg	10
8	input	v	m/s	0.1
9	output	E	J	5.00E-02
10	output	k	N/m	1.02E+03
11	output	xmax-elas	m	0.00988212
12	output	w-elas	rad/s	10.1192885
13	output	w-elas	hz	1.61053479
14	output	t/2-elas	s	0.31045588
15	output	amax-elas	m/s <sup>2</sup>	1.01192885
16	output	Fmax-elas	N	10.1192885
17	output	smax-elas	Pa	1.90E+08



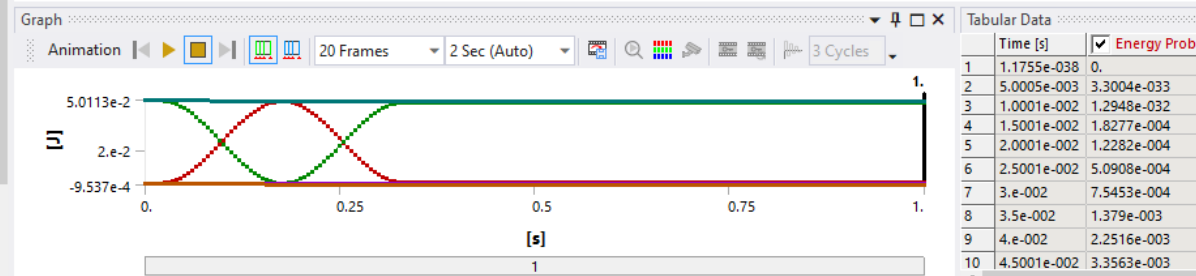
# • Resultados

Con tan poca energía el impacto ha sido elástico y tiene un rebote. Con 0.3[s] tendríamos suficiente.



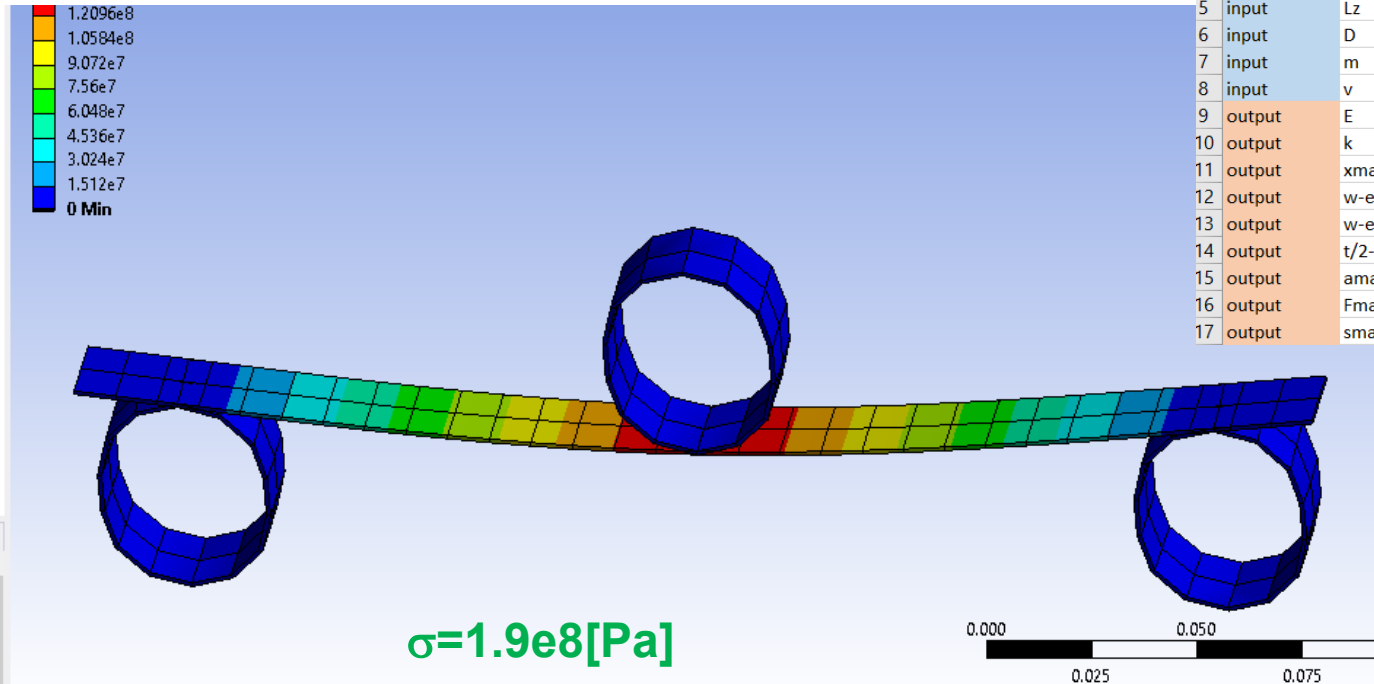
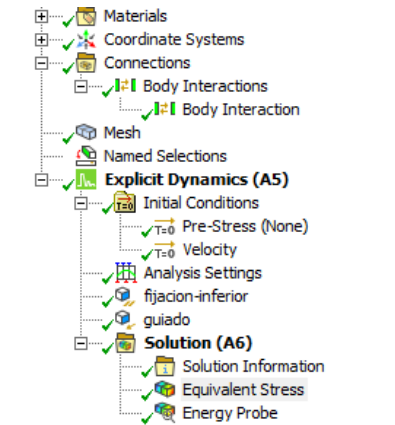
	A	B	C	D
1	input	E	Pa	2.00E+11
2	input	nu	-	3.00E-01
3	input	Lx	m	0.3
4	input	Ly	m	0.02
5	input	Lz	m	0.001
6	input	D	m	0.25
7	input	m	kg	10
8	input	v	m/s	0.1
9	output	E	J	5.00E-02
10	output	k	N/m	1.02E+03
11	output	xmax-elas	m	0.00988212
12	output	w-elas	rad/s	10.1192885
13	output	t-elas	hz	1.61053479
14	output	t/2-elas	s	0.31045588
15	output	amax-elas	m/s <sup>2</sup>	1.01192885
16	output	Fmax-elas	N	10.1192885
17	output	smax-elas	Pa	1.90E+08

Details of "Energy Probe"	
Definition	
Type	Energy
Geometry	System Energy
Suppressed	No
Options	
Result Selection	All
Display Time	End Time
Results	
Maximum Value Over Time	
Internal	4.9461e-002 J
Kinetic	5.0113e-002 J
Plastic Work	0. J
Hourglass	3.7093e-005 J
Contact	0. J
Total	5.0113e-002 J
Minimum Value Over Time	
Internal	0. J
Kinetic	6.6202e-006 J



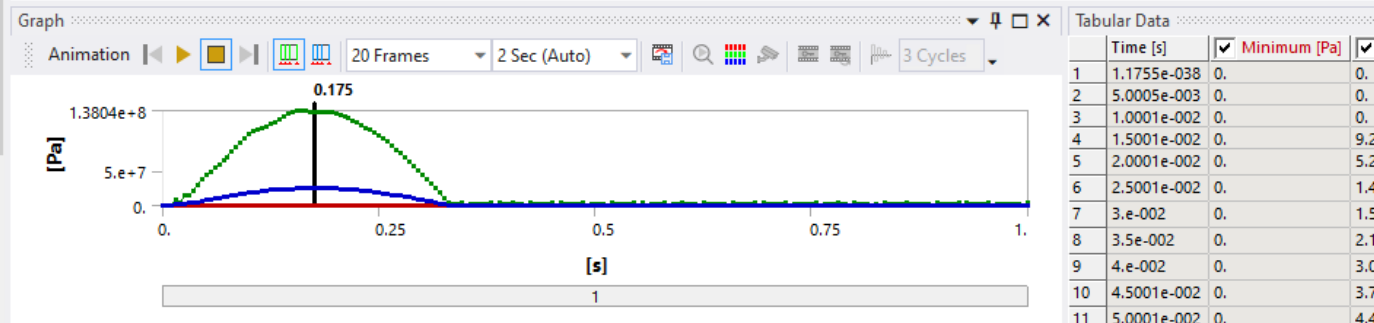
# Resultados

La máxima tensión es de unos 138[MPa] menor que 200[MPa]



details of "Equivalent Stress"

Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Position	Top/Bottom
Definition	
Type	Equivalent (von-Mises) ...
By	Time
<input type="checkbox"/> Display Time	0.175 s
Calculate Time History	Yes
Identifier	
Suppressed	No
Integration Point Results	
Display Option	Averaged
Average Across Bodies	No
Results	
<input type="checkbox"/> Minimum	0. Pa
<input type="checkbox"/> Maximum	1.3608e+008 Pa
<input type="checkbox"/> Average	2.5427e+007 Pa

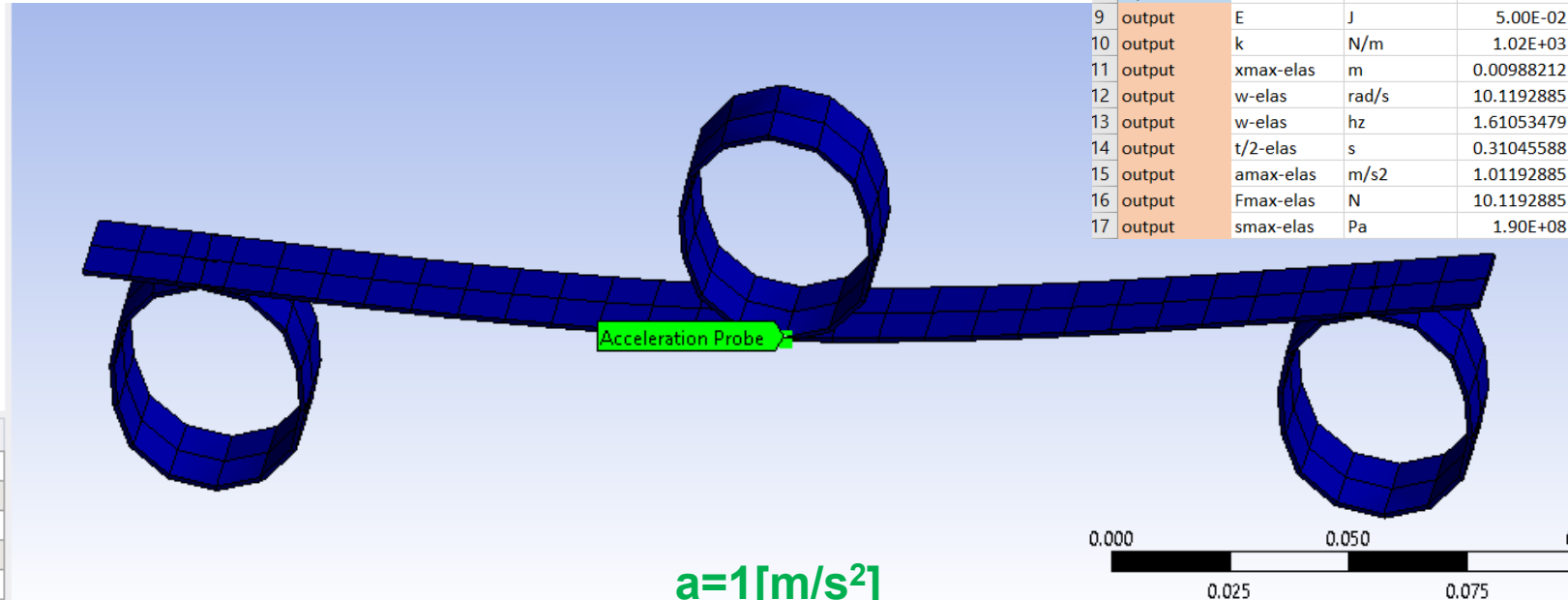
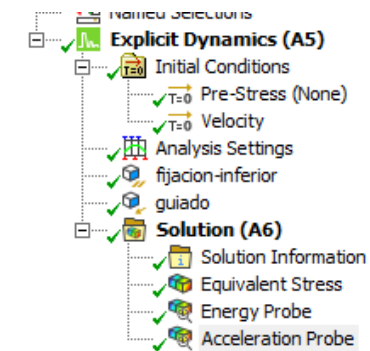


	A	B	C	D
1	input	E	Pa	2.00E+11
2	input	nu	-	3.00E-01
3	input	Lx	m	0.3
4	input	Ly	m	0.02
5	input	Lz	m	0.001
6	input	D	m	0.25
7	input	m	kg	10
8	input	v	m/s	0.1
9	output	E	J	5.00E-02
10	output	k	N/m	1.02E+03
11	output	xmax-elas	m	0.00988212
12	output	w-elas	rad/s	10.1192885
13	output	w-elas	hz	1.61053479
14	output	t/2-elas	s	0.31045588
15	output	amax-elas	m/s2	1.01192885
16	output	Fmax-elas	N	10.1192885
17	output	smax-elas	Pa	1.90E+08

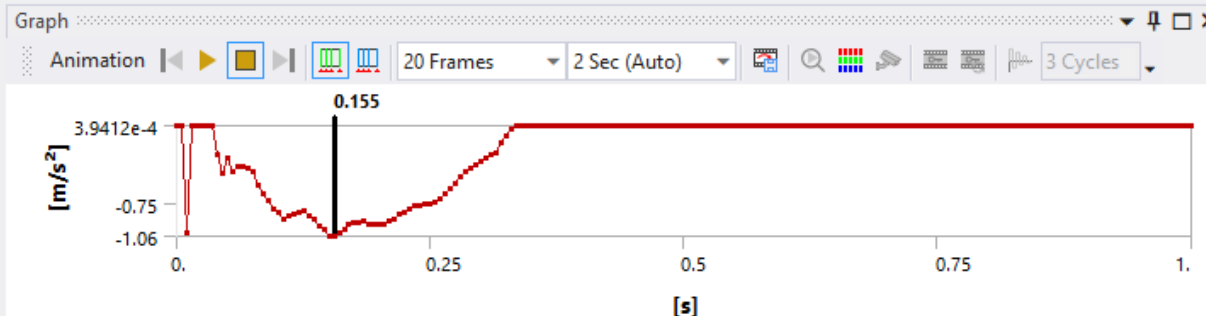
# Resultados

La aceleración es pequeña de  $1\text{ [m/s}^2\text{]}\approx 0.1\text{ [g]}$ .

	A	B	C	D
1	input	E	Pa	2.00E+11
2	input	nu	-	3.00E-01
3	input	Lx	m	0.3
4	input	Ly	m	0.02
5	input	Lz	m	0.001
6	input	D	m	0.25
7	input	m	kg	10
8	input	v	m/s	0.1
9	output	E	J	5.00E-02
10	output	k	N/m	1.02E+03
11	output	xmax-elas	m	0.00988212
12	output	w-elas	rad/s	10.1192885
13	output	w-elas	hz	1.61053479
14	output	t/2-elas	s	0.31045588
15	output	amax-elas	m/s <sup>2</sup>	1.01192885
16	output	Fmax-elas	N	10.1192885
17	output	smax-elas	Pa	1.90E+08



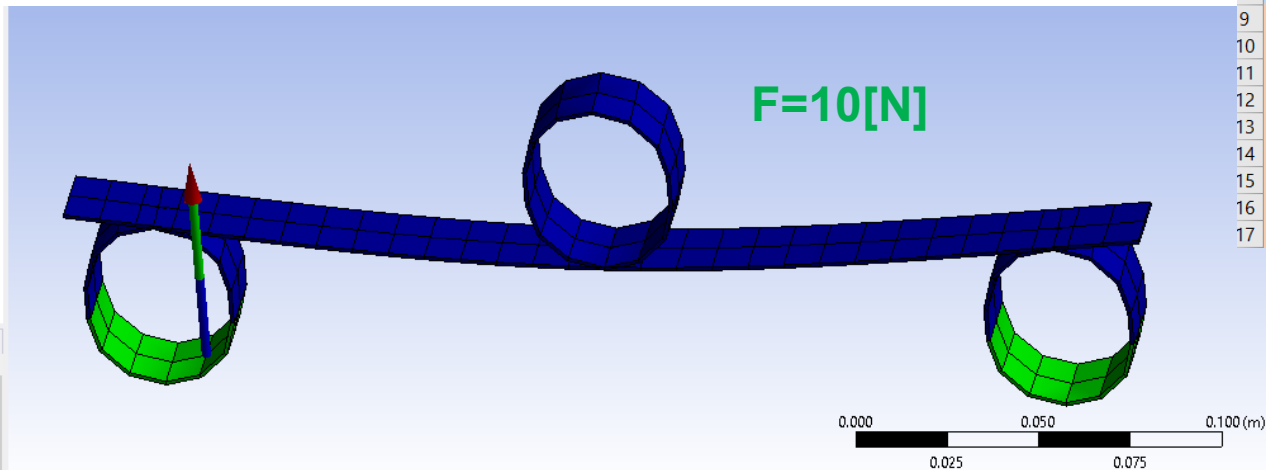
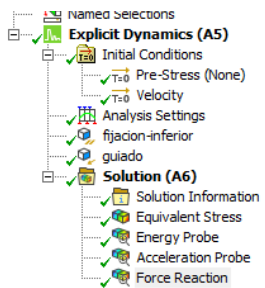
Definition	
Type	Acceleration
Location Method	Geometry Selection
Geometry	1 Vertex
Orientation	Global Coordinate System
Suppressed	No
Options	
Result Selection	Z Axis
Display Time	0.155 s
Spatial Resolution	Use Maximum
Results	
Z Axis	-1.06 m/s <sup>2</sup>
Maximum Value Over Time	
Z Axis	3.9412e-004 m/s <sup>2</sup>
Minimum Value Over Time	
Z Axis	-1.06 m/s <sup>2</sup>
Information	



Tabular Data		
Time [s]	Acceleration Probe (Z)	Acceleration Probe (Z)
1	1.1755e-038	0.
2	5.0005e-003	0.
3	1.0001e-002	-1.0196
4	1.5001e-002	-1.5259e-004
5	2.0001e-002	1.1237e-005
6	2.5001e-002	1.572e-004
7	3.e-002	-2.7125e-004
8	3.5e-002	3.9412e-004
9	4.e-002	-0.27331
10	4.5e-002	0.16181

# • Resultados

La fuerza de reacción equivale a la masa \* aceleración  $\approx 10 \cdot 1 = 10 \text{ [N]}$



	A	B	C	D
1	input	E	Pa	2.00E+11
2	input	nu	-	3.00E-02
3	input	Lx	m	0.3
4	input	Ly	m	0.02
5	input	Lz	m	0.002
6	input	D	m	0.25
7	input	m	kg	10
8	input	v	m/s	0.2
9	output	E	J	5.00E-02
10	output	k	N/m	1.02E+03
11	output	xmax-elas	m	0.0098821
12	output	w-elas	rad/s	10.119288
13	output	w-elas	hz	1.6105347
14	output	t/2-elas	s	0.3104558
15	output	amax-elas	m/s <sup>2</sup>	1.0119288
16	output	Fmax-elas	N	10.119288
17	output	smax-elas	Pa	1.90E+08

Details of "Force Reaction"

Definition	
Type	Force Reaction
Location Method	Boundary Condition
Boundary Condition	fijacion-inferior
Orientation	Global Coordinate System
Suppressed	No

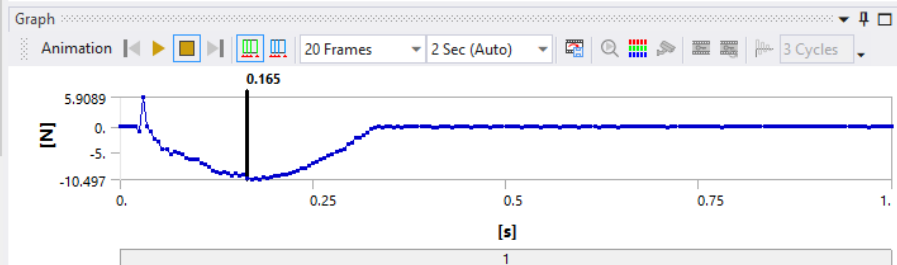
Options	
Result Selection	All
Display Time	0.165 s

Results	
X Axis	5.1387e-002 N
Y Axis	0.10872 N
Z Axis	-10.249 N
Total	10.249 N

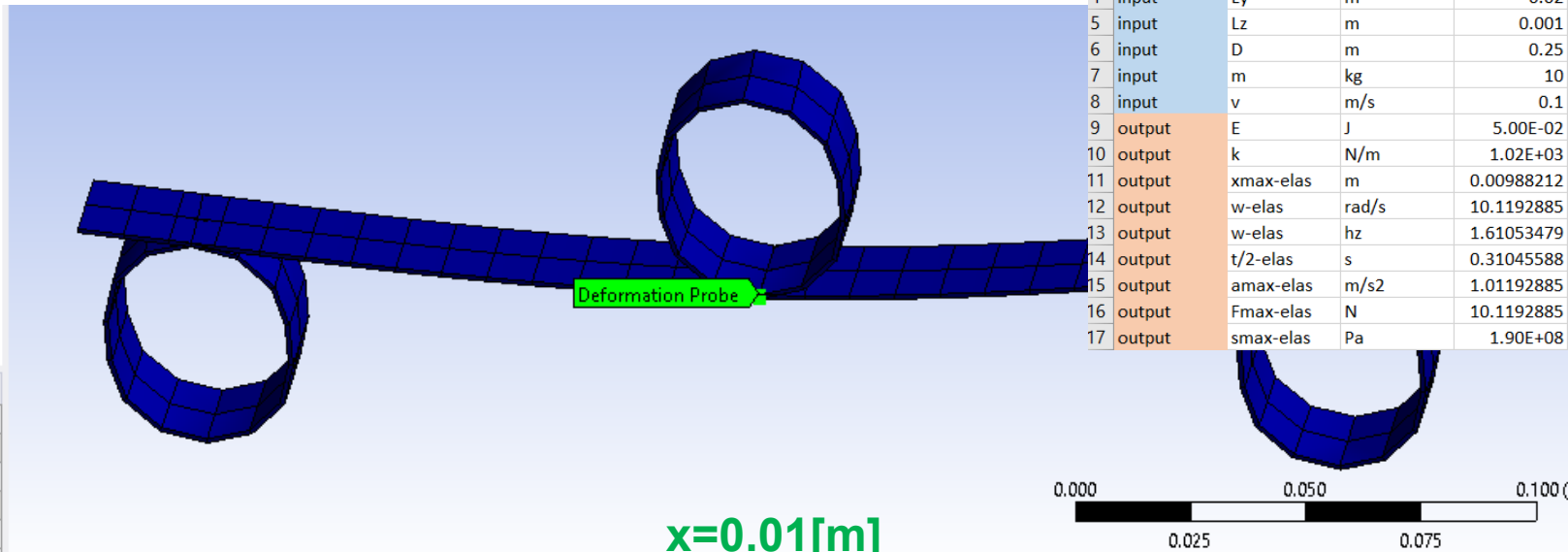
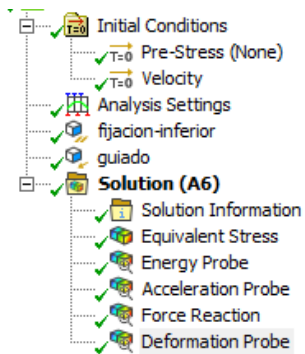
Maximum Value Over Time	
X Axis	0.37587 N
Y Axis	3.8996 N
Z Axis	5.9089 N
Total	10.249 N



Time [s]	Force Reaction (X) [N]	Force Reaction (Y) [N]	Force Reaction (Z) [N]	Force Reaction (Total) [N]
1	1.1755e-038	0.	0.	0.
2	5.0005e-003	0.	0.	0.
3	1.0001e-002	0.	0.	0.
4	1.5001e-002	0.	0.	0.
5	2.0001e-002	0.	0.	0.
6	2.5001e-002	0.24534	3.8996	-0.85942
7	3.e-002	0.37587	1.5493	5.9089
8	3.5e-002	3.3161e-002	-1.5813	-6.2428e-003
9	4.e-002	-0.21275	-0.53407	-0.82803
10	4.5001e-002	-0.42602	0.93114	-2.5006
11	5.0001e-002	0.3414	0.18776	-2.9584

# • Resultados

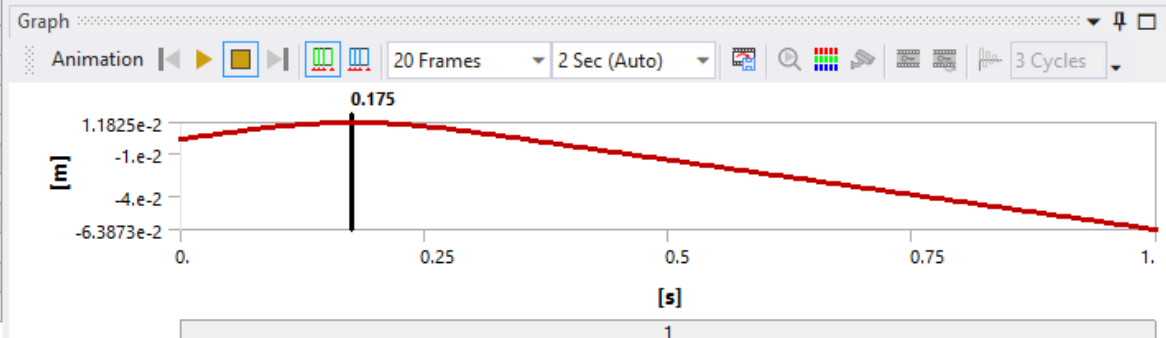
El desplazamiento lo podríamos encontrar como la doble integral de la aceleración y da unos 10[mm] de acuerdo con la rigidez.



	A	B	C	D
1	input	E	Pa	2.00E+11
2	input	nu	-	3.00E-01
3	input	Lx	m	0.3
4	input	Ly	m	0.02
5	input	Lz	m	0.001
6	input	D	m	0.25
7	input	m	kg	10
8	input	v	m/s	0.1
9	output	E	J	5.00E-02
10	output	k	N/m	1.02E+03
11	output	xmax-elas	m	0.00988212
12	output	w-elas	rad/s	10.1192885
13	output	w-elas	hz	1.61053479
14	output	t/2-elas	s	0.31045588
15	output	amax-elas	m/s <sup>2</sup>	1.01192885
16	output	Fmax-elas	N	10.1192885
17	output	smax-elas	Pa	1.90E+08

Details of "Deformation Probe" ▾ □ ×

Definition	
Type	Deformation
Location Method	Geometry Selection
Geometry	1 Vertex
Orientation	Global Coordinate System
Suppressed	No
Options	
Result Selection	Z Axis
<input type="checkbox"/> Display Time	0.175 s
Spatial Resolution	Use Maximum
Results	
<input type="checkbox"/> Z Axis	1.1825e-002 m
Maximum Value Over Time	
<input type="checkbox"/> Z Axis	1.1825e-002 m
Minimum Value Over Time	
<input type="checkbox"/> Z Axis	-6.3873e-002 m
Information	



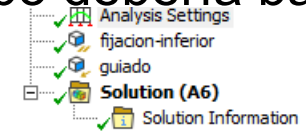
Tabular Data ▾ □ ×

	Time [s]	Deformation Probe (Z) [m]
1	1.1755e-038	0.
2	5.0005e-003	5.0005e-004
3	1.0001e-002	1.0001e-003
4	1.5001e-002	1.4987e-003
5	2.0001e-002	1.9962e-003
6	2.5001e-002	2.4927e-003
7	3.e-002	2.9877e-003
8	3.5e-002	3.4803e-003
9	4.e-002	3.969e-003
10	4.5001e-002	4.4527e-003
11	5.0001e-002	4.93e-003

## • Mass scaling

Como el cálculo tarda mucho vamos a hacer “trampas peligrosas” y escalar la masa para que el time step pase de  $1.5e-6$  a  $1e-5$ [s] con lo que la masa de cada pieza se aumentará  $x(1e-5/1.5e-6)^2 \approx 40$  y tendremos 100000 pasos y el tiempo debería bajar.

Vemos que el resultado se puede ver alterado pues la probeta pasará a pesar 1.6[kg] no despreciable respecto a los 10[kg] del impactador que se le añadirán 0.8[kg]



Details of "probeta"	
Coordinate System	Default Coordinate S
Reference Temperature	By Environment
<input type="checkbox"/> Thickness	1.e-003 m
Thickness Mode	Manual
Offset Type	Middle
<b>Material</b>	
<input type="checkbox"/> Assignment	yield200
<b>Bounding Box</b>	
<b>Properties</b>	
<input type="checkbox"/> Volume	6.e-006 m <sup>3</sup>
<input type="checkbox"/> Mass	4.71e-002 kg
Centroid X	0. m
Centroid Y	0. m
Centroid Z	0. m
<input type="checkbox"/> Moment of Inertia Ip1	1.57e-006 kg·m <sup>2</sup>
<input type="checkbox"/> Moment of Inertia Ip2	3.5325e-004 kg·m <sup>2</sup>
<input type="checkbox"/> Moment of Inertia Ip3	3.5482e-004 kg·m <sup>2</sup>
<input type="checkbox"/> Surface Area(approx.)	6.e-003 m <sup>2</sup>

Details of "impactador"	
<b>Graphics Properties</b>	
<b>Definition</b>	
<input type="checkbox"/> Suppressed	No
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate S.
Reference Temperature	By Environment
<input type="checkbox"/> Thickness	1.e-003 m
Thickness Mode	Manual
Offset Type	Middle
<b>Material</b>	
<input type="checkbox"/> Assignment	Structural Steel
<b>Bounding Box</b>	
<b>Properties</b>	
<input type="checkbox"/> Volume	2.9123e-006 m <sup>3</sup>
<input type="checkbox"/> Mass	2.2861e-002 kg
Centroid X	-3.9818e-017 m
Centroid Y	-5.8556e-020 m
Centroid Z	-2.6e-002 m

Details of "Analysis Settings"	
Number Of Steps	1
Current Step Number	1
Load Step Type	Explicit Time Integratio
End Time	1.
Resume From Cycle	0
Maximum Number of Cycles	1e+07
Maximum Energy Error	0.1
Reference Energy Cycle	0
Initial Time Step	1.e-005 s
Minimum Time Step	1.e-005 s
Maximum Time Step	1.e-004 s
Time Step Safety Factor	0.9
Characteristic Dimension	Opposing Faces
Automatic Mass Scaling	Yes
Minimum CFL Time Step	1.e-004 s
Maximum Element Scaling	1000.
Maximum Part Scaling	1000.
Update Frequency	0

# • Resultados

Ahora ha ido mucho más rápido pero hay que vigilar las subidas de masa

```
Cycle: 21263, Time: 1.000E+00s, Time Inc.: 4.705E-05s, Progress: 100.00%, Est. Clock Time Remaining: 0s  
Cycle: 21264, Time: 1.000E+00s, Time Inc.: 4.705E-05s, Progress: 100.00%, Est. Clock Time Remaining: -
```

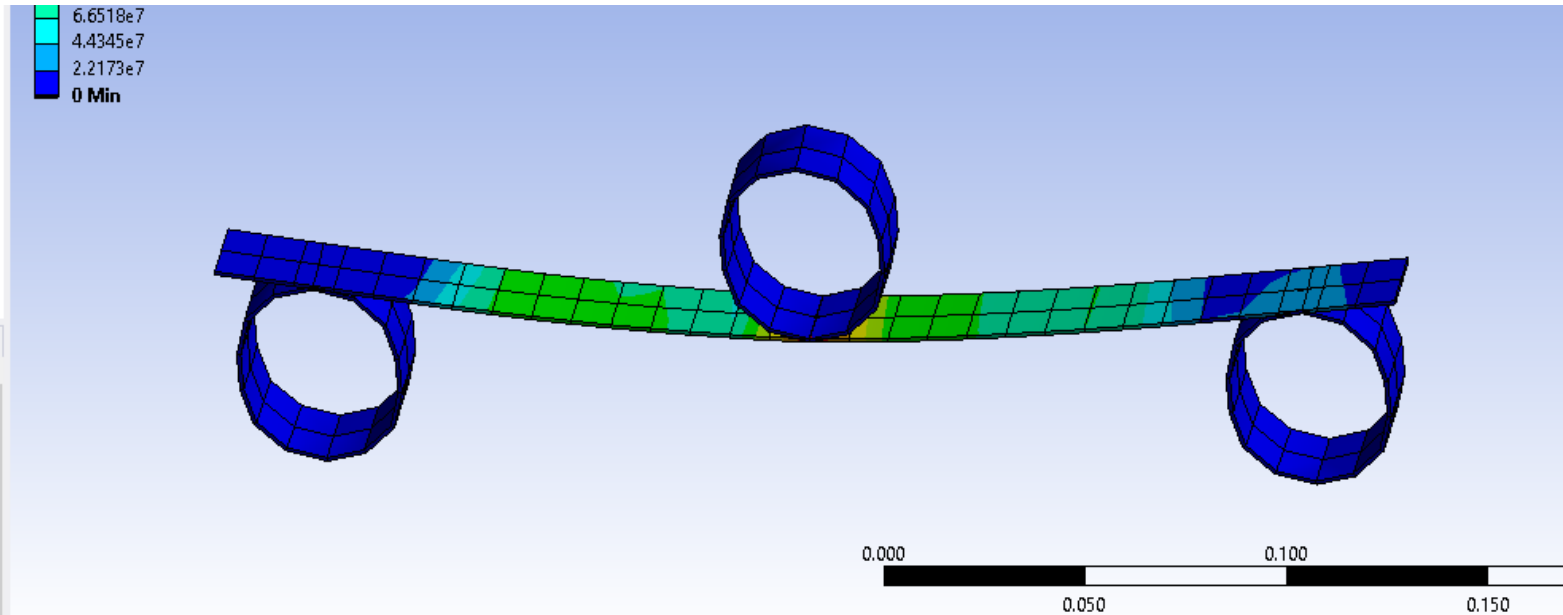
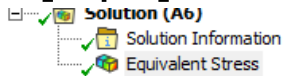
## SIMULATION ELAPSED TIME SUMMARY


```
EXECUTION FROM CYCLE      0 TO      21264  
ELAPSED RUN TIME IN SOLVER =      2.21000E-01 Minutes  
TOTAL ELAPSED RUN TIME   =      2.51217E-01 Minutes  
JOB RAN OVER      2 WORKERS  
JOB RAN USING Intel MPI  
JOB RAN USING DECOMPOSITION AUTO
```

```
Problem terminated .... wrapup time reached
```

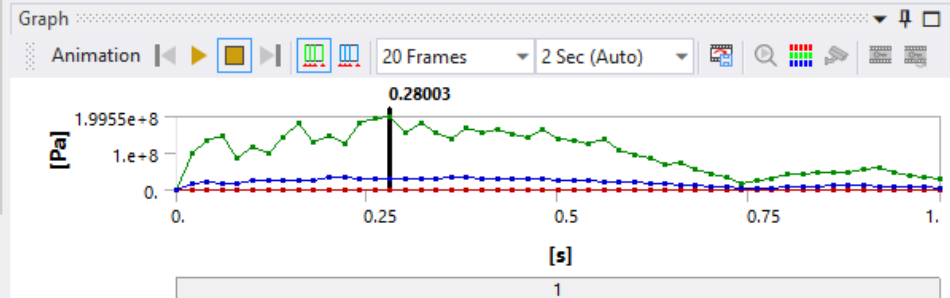
# • Resultados


Parece ser que la subida de masa hace que plastifique pasando de 138[MPa] a 200[MPa]



Details of "Equivalent Stress" 

Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Position	Top/Bottom
Definition	
Type	Equivalent (von-Mises) Stress
By	Time
<input type="checkbox"/> Display Time	0.28003 s
Calculate Time History	Yes
Identifier	
Suppressed	No
Integration Point Results	
Display Option	Averaged
Average Across Bodies	No
Results	
<input type="checkbox"/> Minimum	0. Pa
<input type="checkbox"/> Maximum	1.9955e+008 Pa
<input type="checkbox"/> Average	3.0366e+007 Pa
Minimum Occurs On	soporte-derecha
Maximum Occurs On	probeta
Minimum Value Over Time	



Tabular Data 

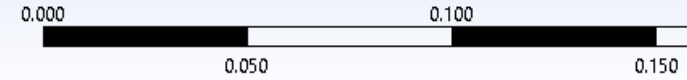
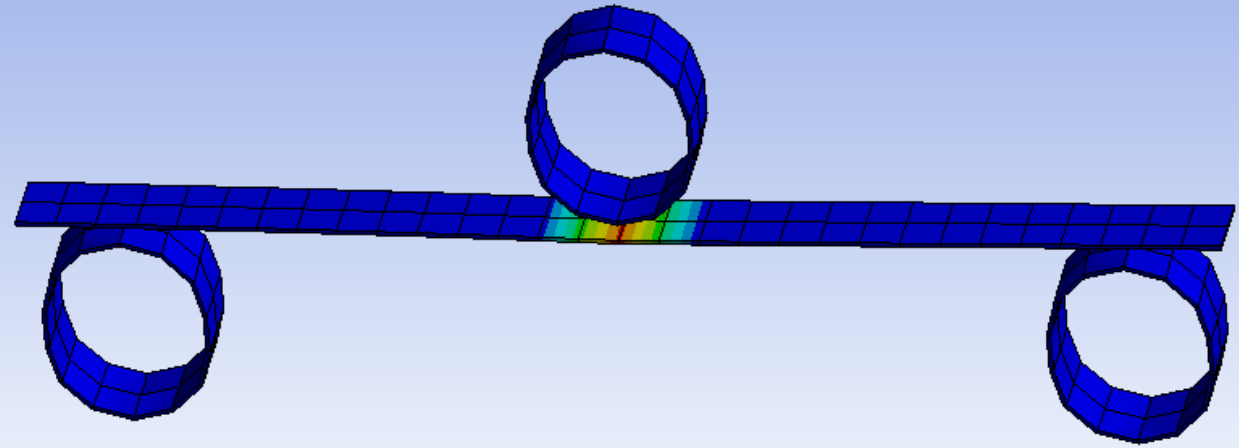
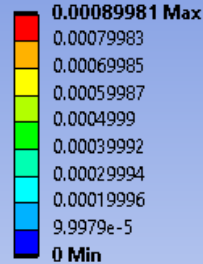
Time [s]	Minimum [Pa]	Maximum [Pa]	Average [Pa]
1	1.1755e-038	0.	0.
2	2.0013e-002	0.	1.0157e+008
3	4.0008e-002	0.	1.3554e+008
4	6.0002e-002	0.	1.4622e+008
5	8.0044e-002	0.	8.6385e+007
6	0.10004	0.	1.1901e+008
7	0.12003	0.	9.9445e+007
8	0.14003	0.	1.4379e+008
9	0.16002	0.	1.8316e+008
10	0.18002	0.	1.5180e+008



**Project\***

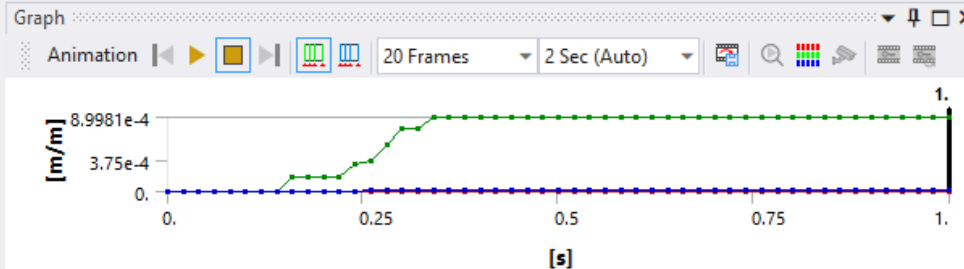
- Model (A4)
  - Geometry
  - Materials
  - Coordinate Systems
  - Connections
  - Mesh
  - Named Selections
  - Explicit Dynamics (A5)
    - Initial Conditions
    - Analysis Settings
      - fijacion-inferior
      - guiado
  - Solution (A6)
    - Solution Information
    - Equivalent Stress
    - Equivalent Plastic Strain

Equivalent Plastic Strain  
 Type: Equivalent Plastic Strain - Top/Bottom  
 Unit: m/m  
 Time: 1.  
 Cycle Number: 21264  
 03/09/2021 15:30



Details of "Equivalent Plastic Strain"

Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Position	Top/Bottom
Definition	
Type	Equivalent Plastic Strain
By	Time
<input type="checkbox"/> Display Time	Last
Calculate Time History	Yes
Identifier	
Suppressed	No
Integration Point Results	
Display Option	Averaged
Average Across Bodies	No
Results	
<input type="checkbox"/> Minimum	0. m/m
<input type="checkbox"/> Maximum	8.9981e-004 m/m
<input type="checkbox"/> Average	2.4855e-005 m/m
Minimum Occurs On	soporte-derecha
Maximum Occurs On	probeta

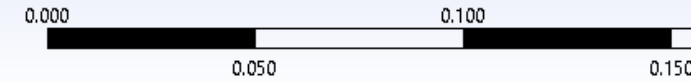
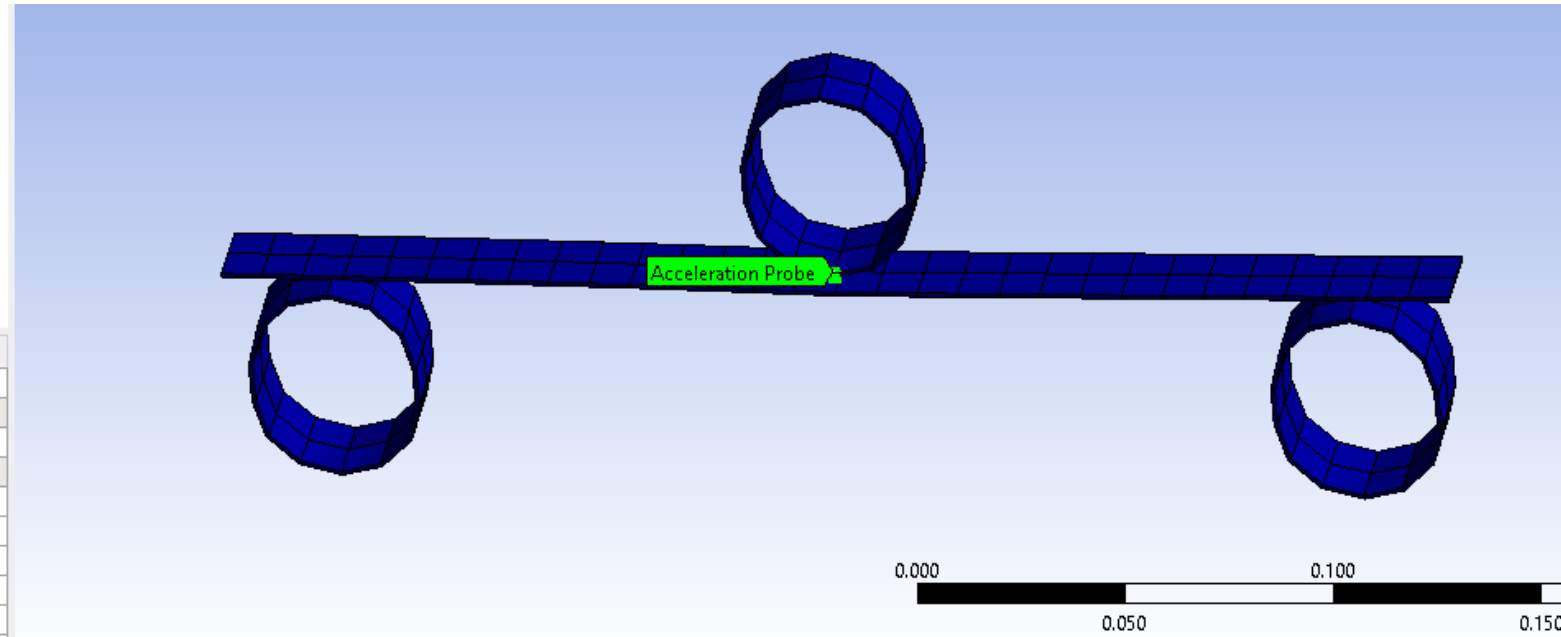
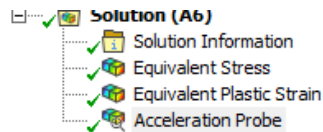


Tabular Data

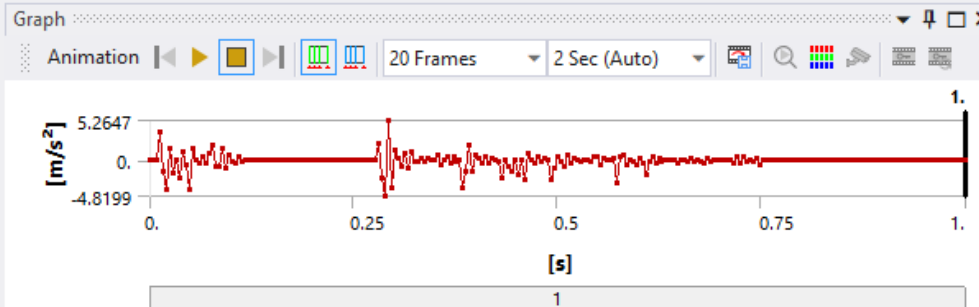
Time [s]	Minimum [m/m]	Maximum [m/m]	Average
1 1.1755e-038	0.	0.	0.
2 2.0013e-002	0.	0.	0.
3 4.0008e-002	0.	0.	0.
4 6.0002e-002	0.	0.	0.
5 8.0044e-002	0.	0.	0.
6 0.10004	0.	0.	0.
7 0.12003	0.	0.	0.
8 0.14003	0.	0.	0.

# • Resultados

La aceleración pasa de  $1[m/s^2]$  a  $-5[m/s^2]$  con forma irregular



Definition	
Type	Acceleration
Location Method	Geometry Selection
Geometry	1 Vertex
Orientation	Global Coordinate System
Suppressed	No
Options	
Result Selection	Z Axis
<input type="checkbox"/> Display Time	End Time
Spatial Resolution	Use Maximum
Results	
Maximum Value Over Time	
<input type="checkbox"/> Z Axis	5.2647 m/s <sup>2</sup>
Minimum Value Over Time	
<input type="checkbox"/> Z Axis	-4.8199 m/s <sup>2</sup>
Information	

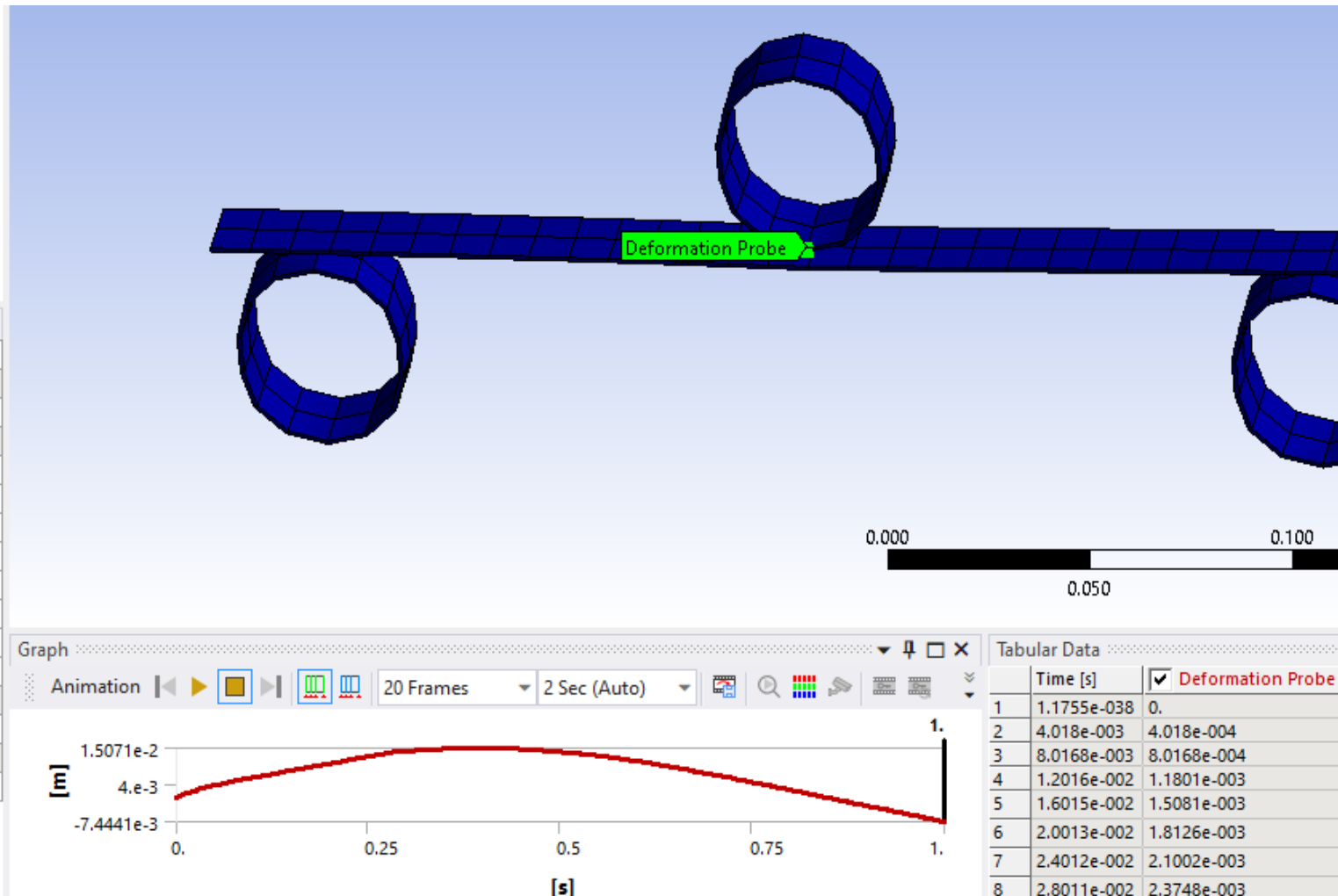


Tabular Data		
Time [s]	Acceleration Probe (Z) [m/s <sup>2</sup> ]	
1	1.1755e-038	0.
2	4.018e-003	0.
3	8.0168e-003	0.
4	1.2016e-002	3.7837
5	1.6015e-002	-1.4663
6	2.0013e-002	-3.9395
7	2.4012e-002	1.5686
8	2.8011e-002	-1.7938
9	3.201e-002	-0.10056

# • Resultados

El desplazamiento pasa de 10[mm] a 15[mm] con forma irregular

- Solution Information
- Equivalent Stress
- Equivalent Plastic Strain
- Acceleration Probe
- Deformation Probe

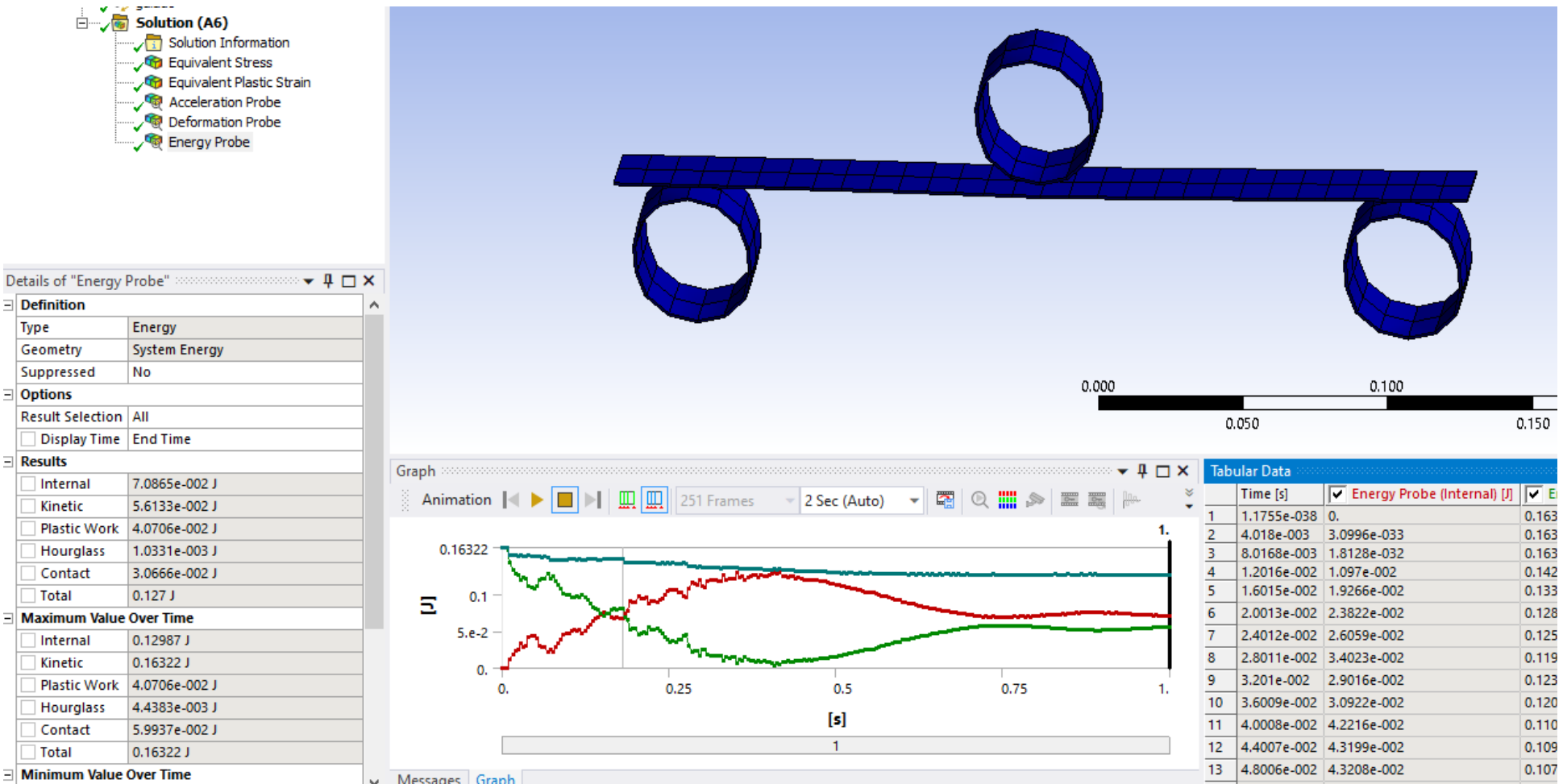


Details of "Deformation Probe" ⌵ □ ×

Definition	
Type	Deformation
Location Method	Geometry Selection
Geometry	1 Vertex
Orientation	Global Coordinate System
Suppressed	No
Options	
Result Selection	Z Axis
<input type="checkbox"/> Display Time	End Time
Spatial Resolution	Use Maximum
Results	
Maximum Value Over Time	
<input type="checkbox"/> Z Axis	1.5071e-002 m
Minimum Value Over Time	
<input type="checkbox"/> Z Axis	-7.4441e-003 m
Information	

# • Resultados

La curva más importante es la de energía donde se observa el riesgo de incrementar la masa pasando de 0.05[J] a 0.16[J] (triplicando energía)



## • Ejercicio

Hacer una simulación para espesor 3[mm] en el que tengamos un impacto elástico de unos 10[mm] con velocidad inicial de 1[m/s<sup>2</sup>].

Ahora necesitamos menos tiempo de cálculo:

```
Cycle: 53774, Time: 8.000E-02s, Time Inc.: 1.488E-06s, Progress: 100
Cycle: 53775, Time: 8.000E-02s, Time Inc.: 1.488E-06s, Progress: 100
```

### SIMULATION ELAPSED TIME SUMMARY

```
EXECUTION FROM CYCLE      0 TO      53775
ELAPSED RUN TIME IN SOLVER =      4.46317E-01 Minutes
TOTAL ELAPSED RUN TIME   =      4.76733E-01 Minutes
JOB RAN OVER      2 WORKERS
JOB RAN USING Intel MPI
JOB RAN USING DECOMPOSITION AUTO
```

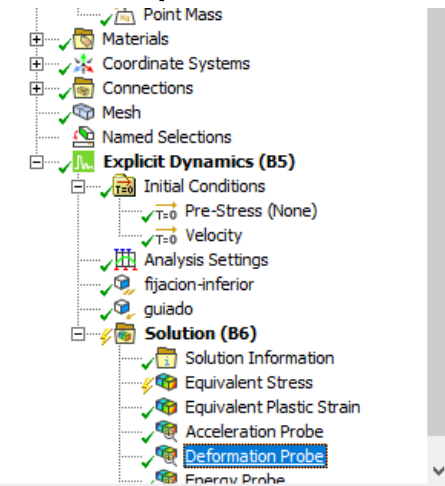
```
Problem terminated .... wrapup time reached
```

```
*****
```

	A	B	C	D	E
1	input	d	kg/m3	7.85E+03	7.85E+03
2	input	E	Pa	2.00E+11	2.00E+11
3	input	nu	-	3.00E-01	3.00E-01
4	input	Lx	m	3.00E-01	3.00E-01
5	input	Ly	m	2.00E-02	2.00E-02
6	input	Lz	m	1.00E-03	3.00E-03
7	input	D	m	2.50E-01	2.50E-01
8	input	m	kg	1.00E+01	3.00E+00
9	input	v	m/s	1.00E-01	1.00E+00
10	input	mesh	m	1.00E-02	1.00E-02
11	input	damp	-	2.00E-01	2.00E-01
12	output	ts-auto	s	1.58E-06	1.58E-06
13	output	E	J	5.00E-02	1.50E+00
14	output	k	N/m	1.02E+03	2.76E+04
15	output	xmax-elas	m	9.88E-03	1.04E-02
16	output	w-elas	rad/s	1.01E+01	9.60E+01
17	output	w-elas	hz	1.61E+00	1.53E+01
18	output	t/2-elas	s	3.10E-01	3.27E-02
19	output	amax-elas	m/s2	1.01E+00	9.60E+01
20	output	Fmax-elas	N	1.01E+01	2.88E+02
21	output	smax-elas	Pa	1.90E+08	6.00E+08
22	input	ts-forced	s	1.00E-05	1.00E-05
23	input	mass-increase	-	3.98E+01	3.98E+01

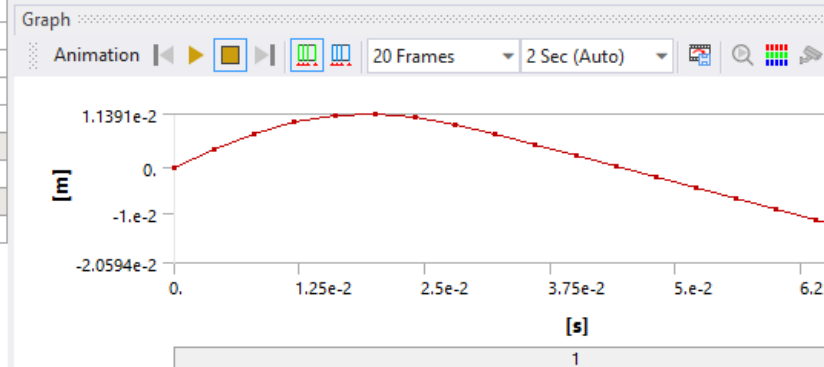
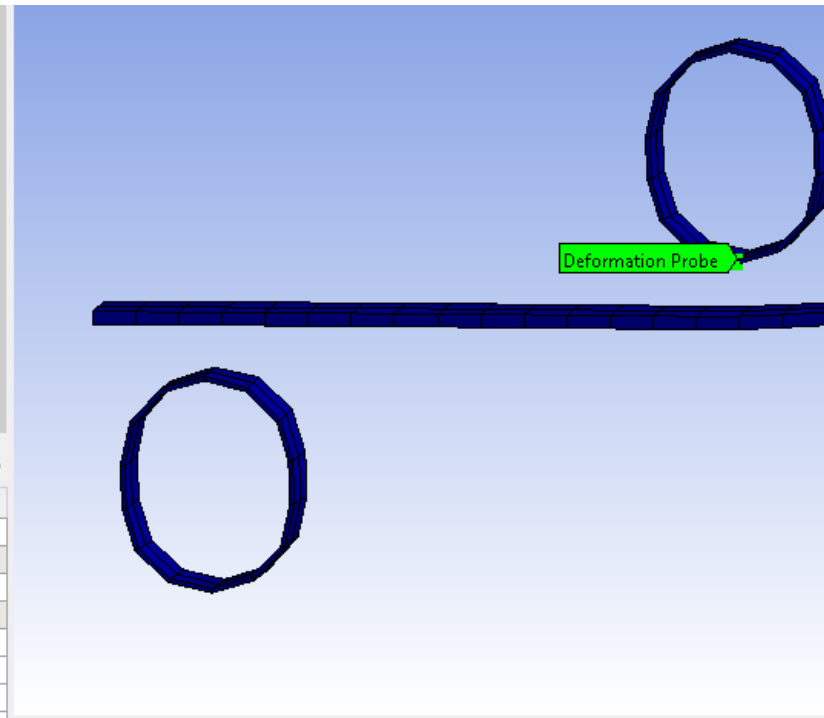
# • Resultados

Desplazamiento de 15[mm] en primera simulación con 3[kg] y se ajusta a 2[kg] para la



Details of "Deformation Probe"

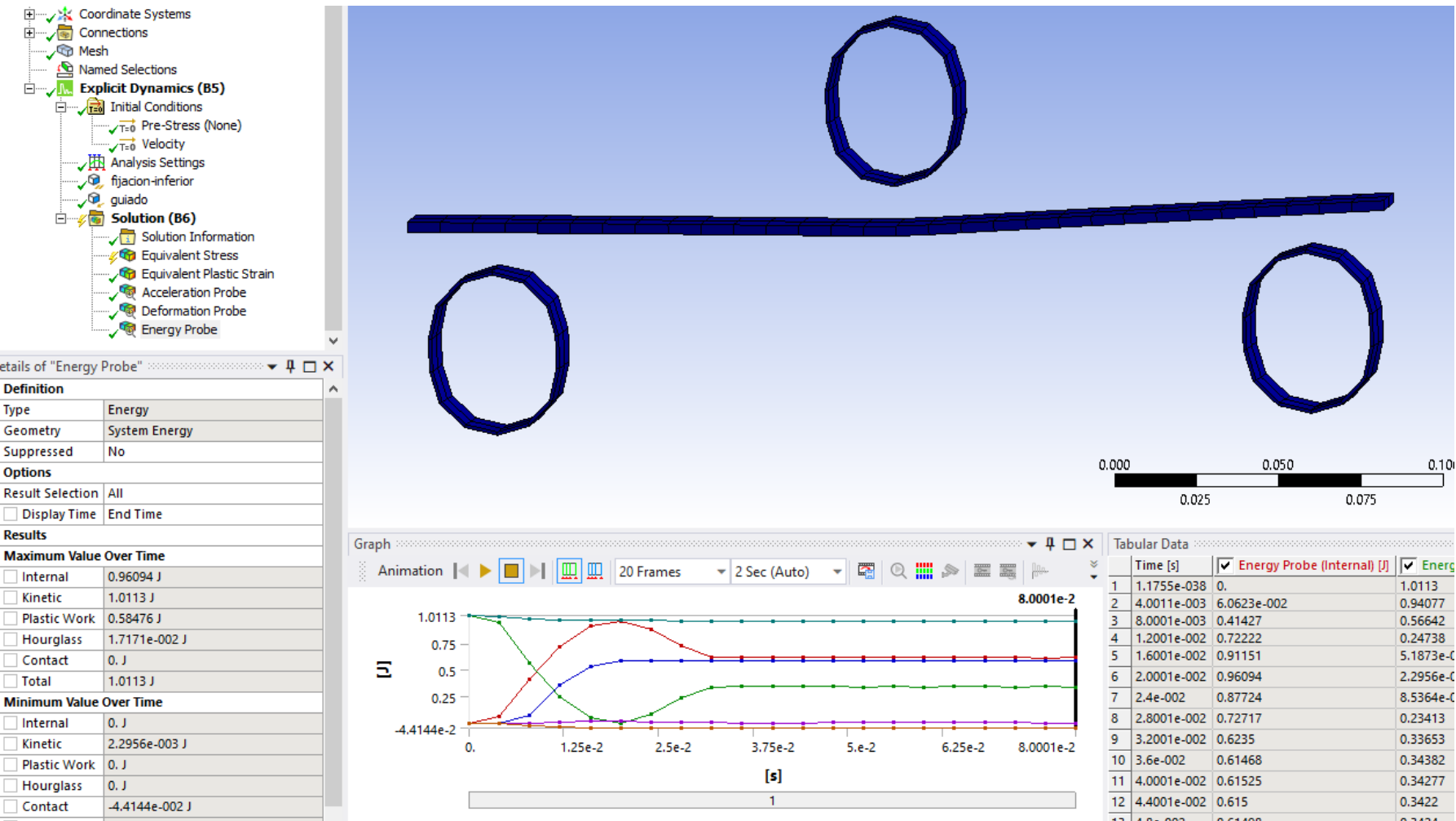
Definition	
Type	Deformation
Location Method	Geometry Selection
Geometry	1 Vertex
Orientation	Global Coordinate System
Suppressed	No
Options	
Result Selection	Z Axis
<input type="checkbox"/> Display Time	End Time
Spatial Resolution	Use Maximum
Results	
Maximum Value Over Time	
<input type="checkbox"/> Z Axis	1.1391e-002 m
Minimum Value Over Time	
<input type="checkbox"/> Z Axis	-2.0594e-002 m
Information	



	A	B	C	D	E
1	input	d	kg/m3	7.85E+03	7.85E+03
2	input	E	Pa	2.00E+11	2.00E+11
3	input	nu	-	3.00E-01	3.00E-01
4	input	Lx	m	3.00E-01	3.00E-01
5	input	Ly	m	2.00E-02	2.00E-02
6	input	Lz	m	1.00E-03	3.00E-03
7	input	D	m	2.50E-01	2.50E-01
8	input	m	kg	1.00E+01	3.00E+00
9	input	v	m/s	1.00E-01	1.00E+00
10	input	mesh	m	1.00E-02	1.00E-02
11	input	damp	-	2.00E-01	2.00E-01
12	output	ts-auto	s	1.58E-06	1.58E-06
13	output	E	J	5.00E-02	1.50E+00
14	output	k	N/m	1.02E+03	2.76E+04
15	output	xmax-elas	m	9.88E-03	1.04E-02
16	output	w-elas	rad/s	1.01E+01	9.60E+01
17	output	w-elas	hz	1.61E+00	1.53E+01
18	output	t/2-elas	s	3.10E-01	3.27E-02
19	output	amax-elas	m/s2	1.01E+00	9.60E+01
20	output	Fmax-elas	N	1.01E+01	2.88E+02
21	output	smax-elas	Pa	1.90E+08	6.00E+08
22	input	ts-forced	s	1.00E-05	1.00E-05
23	input	mass-increase	-	3.98E+01	3.98E+01

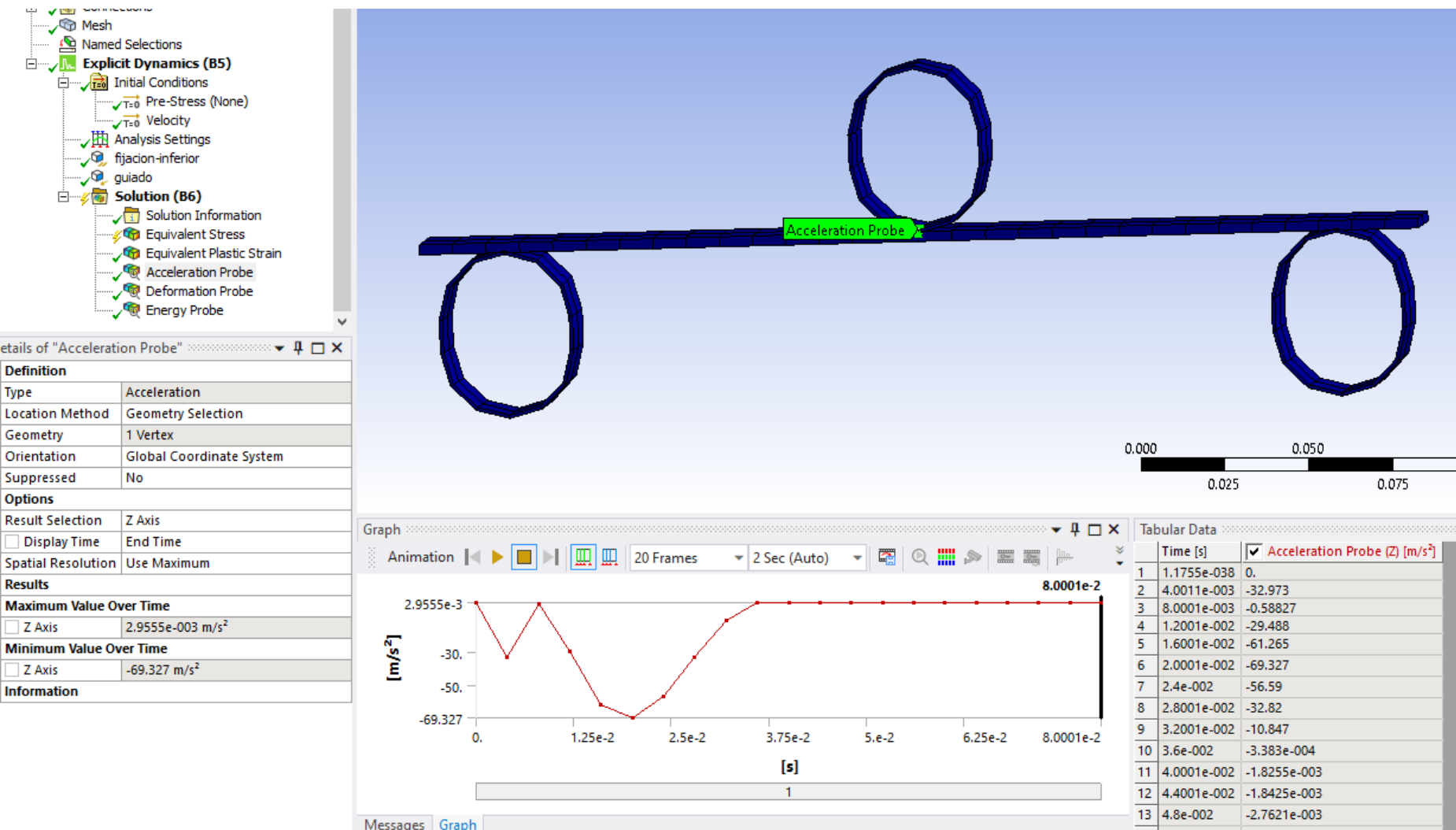
# • Resultados

Comprobamos energías y vemos se respeta bien el tener 1[J]



# • Resultados

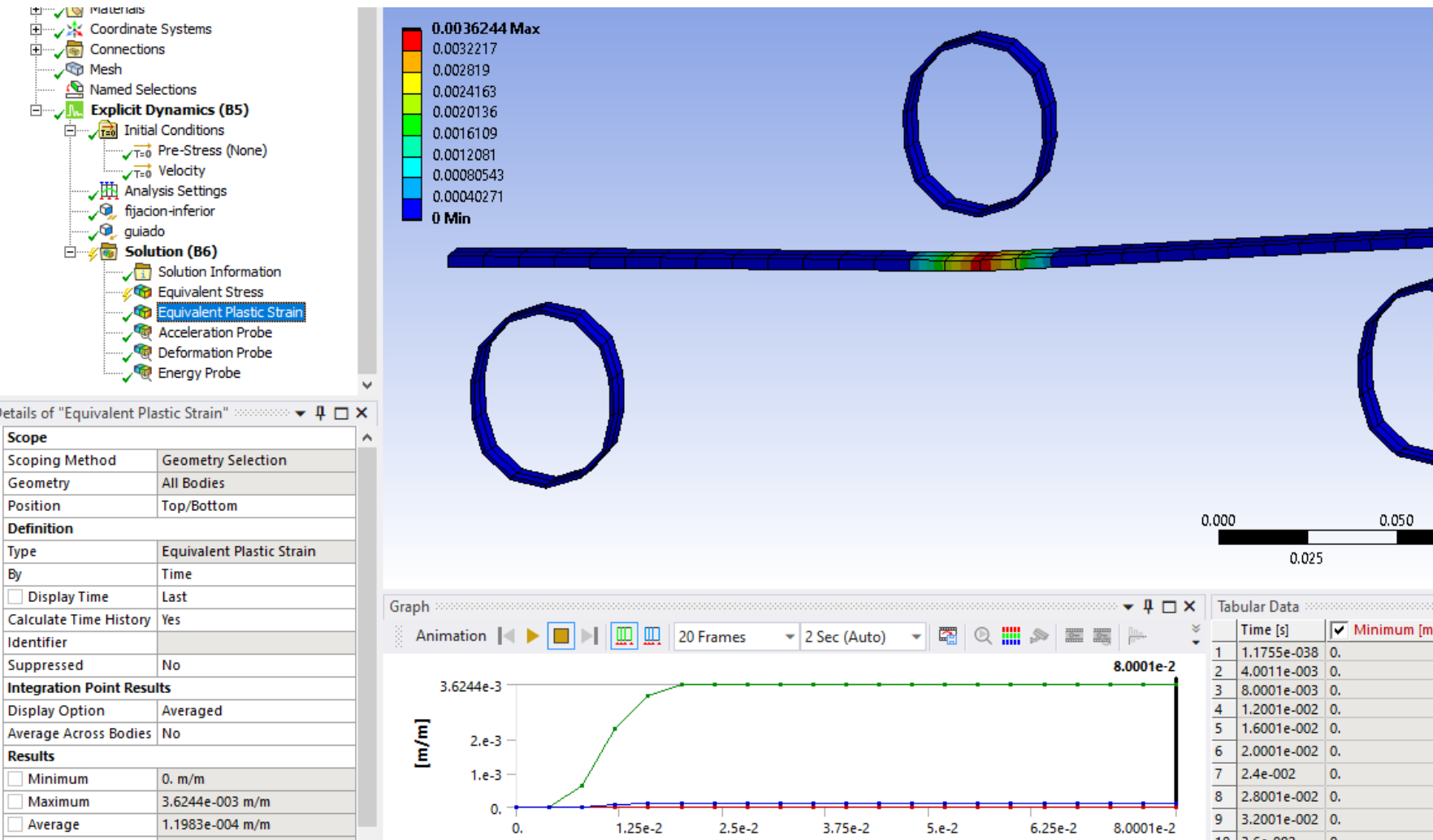
La aceleración llega a unos 7[g]





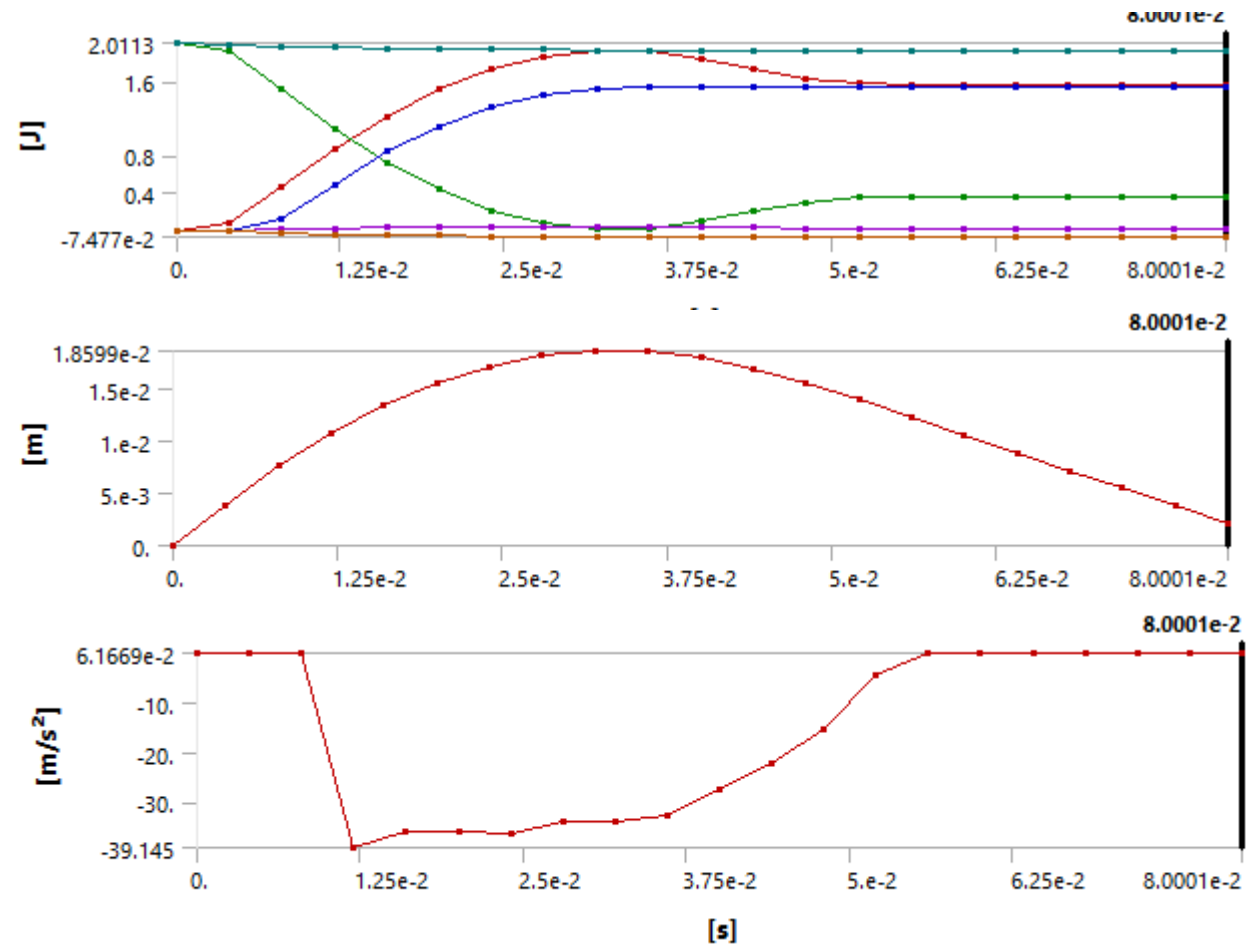
# • Resultados

Hay una ligera zona de deformación plástica.



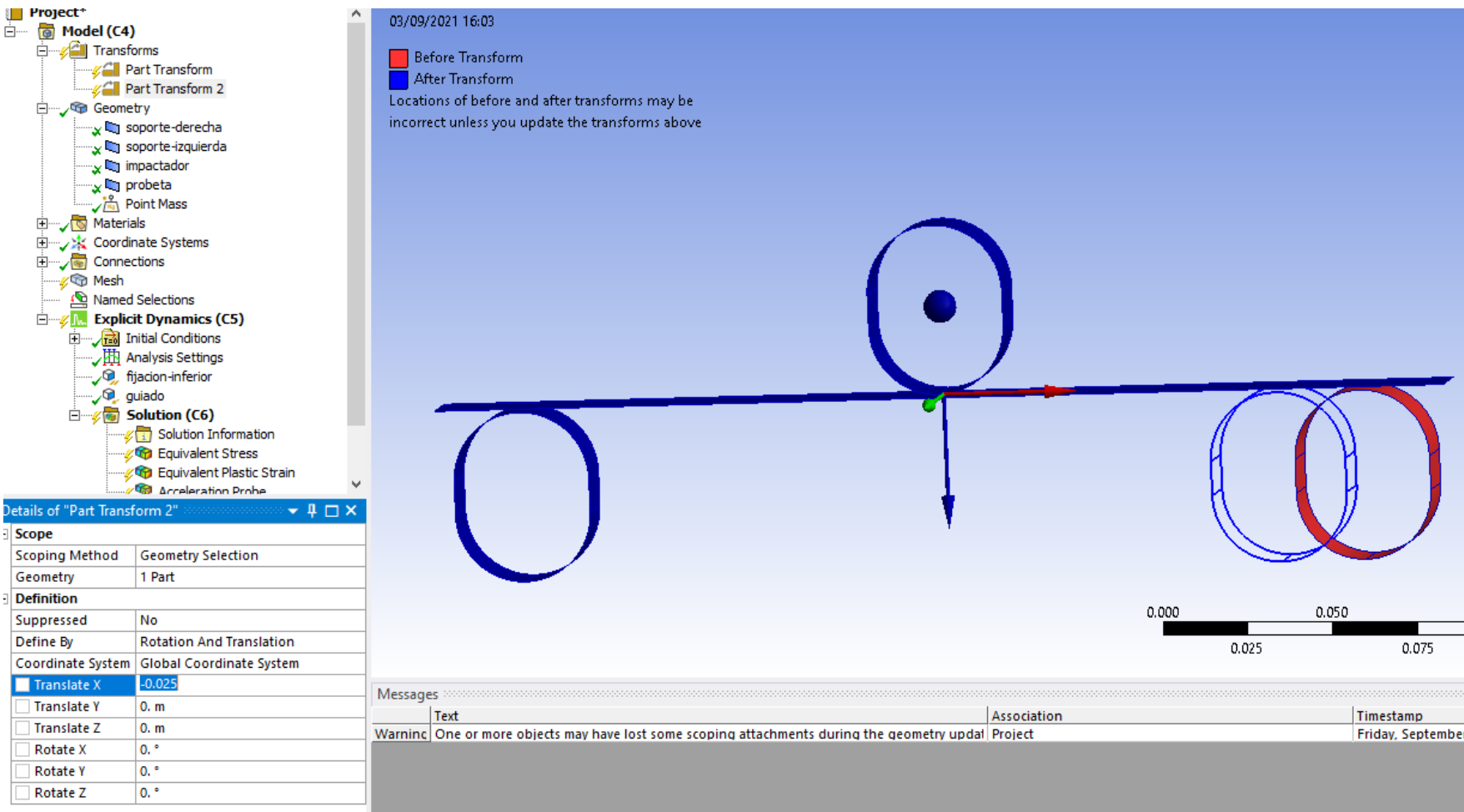
## • Duplicar masa

Si ponemos el doble de masa y energía entonces plastifica más y se pasa de 10[mm] a 18.6[mm] con una aceleración que baja de 7 a 4[g].



# • Juntar apoyos

Si juntamos los apoyos de 250[mm] a 200[mm] será más rígido con  $m=2[\text{kg}]$ ,  $v1=[\text{m/s}]$  y  $t=1[\text{mm}]$



03/09/2021 16:03

Before Transform  
 After Transform  
 Locations of before and after transforms may be incorrect unless you update the transforms above

Project\*

- Model (C4)
  - Transforms
    - Part Transform
    - Part Transform 2
  - Geometry
    - soporte-derecha
    - soporte-izquierda
    - impactador
    - probeta
    - Point Mass
  - Materials
  - Coordinate Systems
  - Connections
  - Mesh
  - Named Selections
  - Explicit Dynamics (C5)
    - Initial Conditions
    - Analysis Settings
      - fijacion-inferior
      - guiado
  - Solution (C6)
    - Solution Information
    - Equivalent Stress
    - Equivalent Plastic Strain
    - Acceleration Probe

Details of "Part Transform 2"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Part
Definition	
Suppressed	No
Define By	Rotation And Translation
Coordinate System	Global Coordinate System
<input checked="" type="checkbox"/> Translate X	-0.025
<input type="checkbox"/> Translate Y	0. m
<input type="checkbox"/> Translate Z	0. m
<input type="checkbox"/> Rotate X	0. °
<input type="checkbox"/> Rotate Y	0. °
<input type="checkbox"/> Rotate Z	0. °

Messages

Text	Association	Timestamp
Warning	One or more objects may have lost some scoping attachments during the geometry updat	Project
		Friday, September

# • Resultados

Si juntamos los apoyos de 250[mm] a 200[mm] será más rígido

	A	B	C	D	E	F
input	d		kg/m3	7.85E+03	7.85E+03	7.85E+03
input	E		Pa	2.00E+11	2.00E+11	2.00E+11
input	nu		-	3.00E-01	3.00E-01	3.00E-01
input	Lx		m	3.00E-01	3.00E-01	3.00E-01
input	Ly		m	2.00E-02	2.00E-02	2.00E-02
input	Lz		m	1.00E-03	3.00E-03	1.00E-03
input	D		m	2.50E-01	2.50E-01	2.00E-01
input	m		kg	1.00E+01	3.00E+00	2.00E+00
input	v		m/s	1.00E-01	1.00E+00	1.00E+00
input	mesh		m	1.00E-02	1.00E-02	1.00E-02
input	damp		-	2.00E-01	2.00E-01	2.00E-01
output	ts-auto		s	1.58E-06	1.58E-06	1.58E-06
output	E		J	5.00E-02	1.50E+00	1.00E+00
output	k		N/m	1.02E+03	2.76E+04	2.00E+03
output	xmax-elas		m	9.88E-03	1.04E-02	3.16E-02
output	w-elas		rad/s	1.01E+01	9.60E+01	3.16E+01
output	w-elas		hz	1.61E+00	1.53E+01	5.03E+00
output	t/2-elas		s	3.10E-01	3.27E-02	9.93E-02
output	amax-elas		m/s2	1.01E+00	9.60E+01	3.16E+01
output	Fmax-elas		N	1.01E+01	2.88E+02	6.32E+01
output	smax-elas		Pa	1.90E+08	6.00E+08	9.49E+08
input	ts-forced		s	1.00E-05	1.00E-05	1.00E-05
input	mass-increase		-	3.98E+01	3.98E+01	3.98E+01

- Resumen.

- Hemos realizado un primer cálculo multipieza con contactos.
- Este cálculo nos sirve para poder validar haciendo ensayos si nuestro material es capaz de absorber energía
- Transformaciones para poder hacer ensayos con diferentes distancias entre apoyos

S04t.- Crash box.

Mejora 2122....

- Repaso última sesión

Cálculo impacto a flexión

Introducción de contacto y fricción

Posibilidades de Mass Scaling

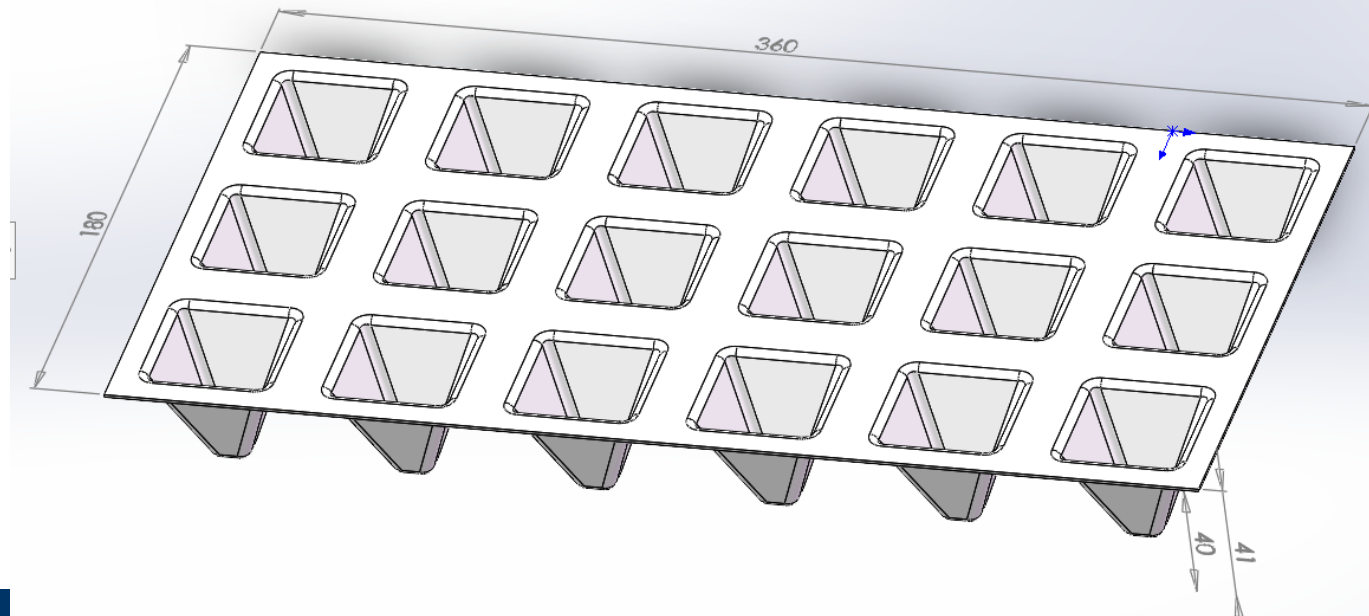
Multipiezas y colocación de dichas piezas

## • Crash box

Vamos a intentar parar en menos de 40[mm] un paquete de 50[kg] que cae con una velocidad de 3.5[m/s]. Como factor de seguridad digamos que usamos sólo 20[mm] por lo que necesitaremos una aceleración de unos 31.2[g] ( $a=v^2/(2s)$ ) en unos 12[ms] ( $t=v/a$ ) con fuerzas de 15.3[kN] ( $F=ma$ )

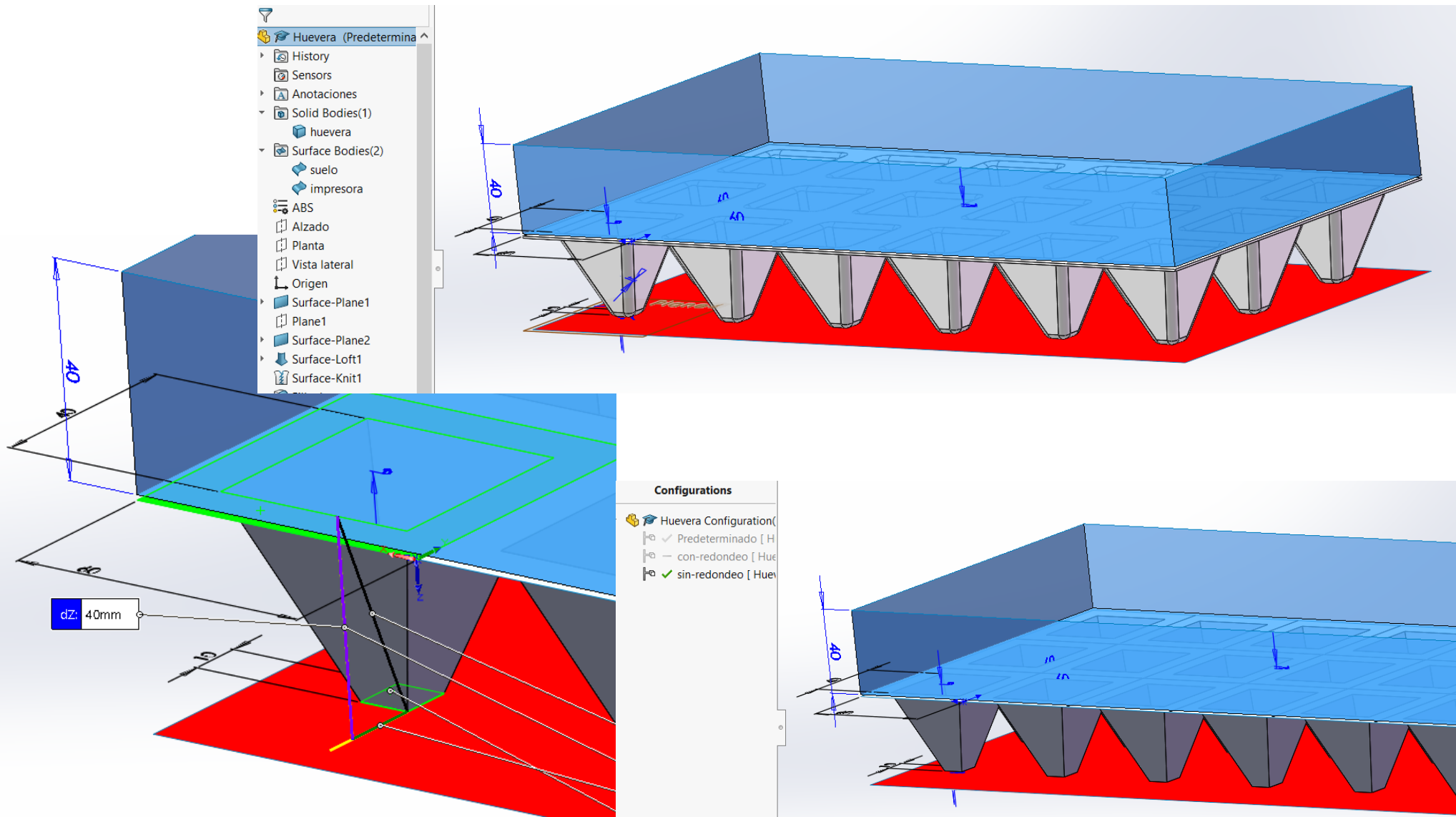
Todos estos números son orientativos pues asumen que la aceleración es constante y seguro no lo va a ser.

La huevera ocupa 6x60=360[mm] x 3x60=180[mm] con los comentados 40[mm] más el espesor de 1[mm].



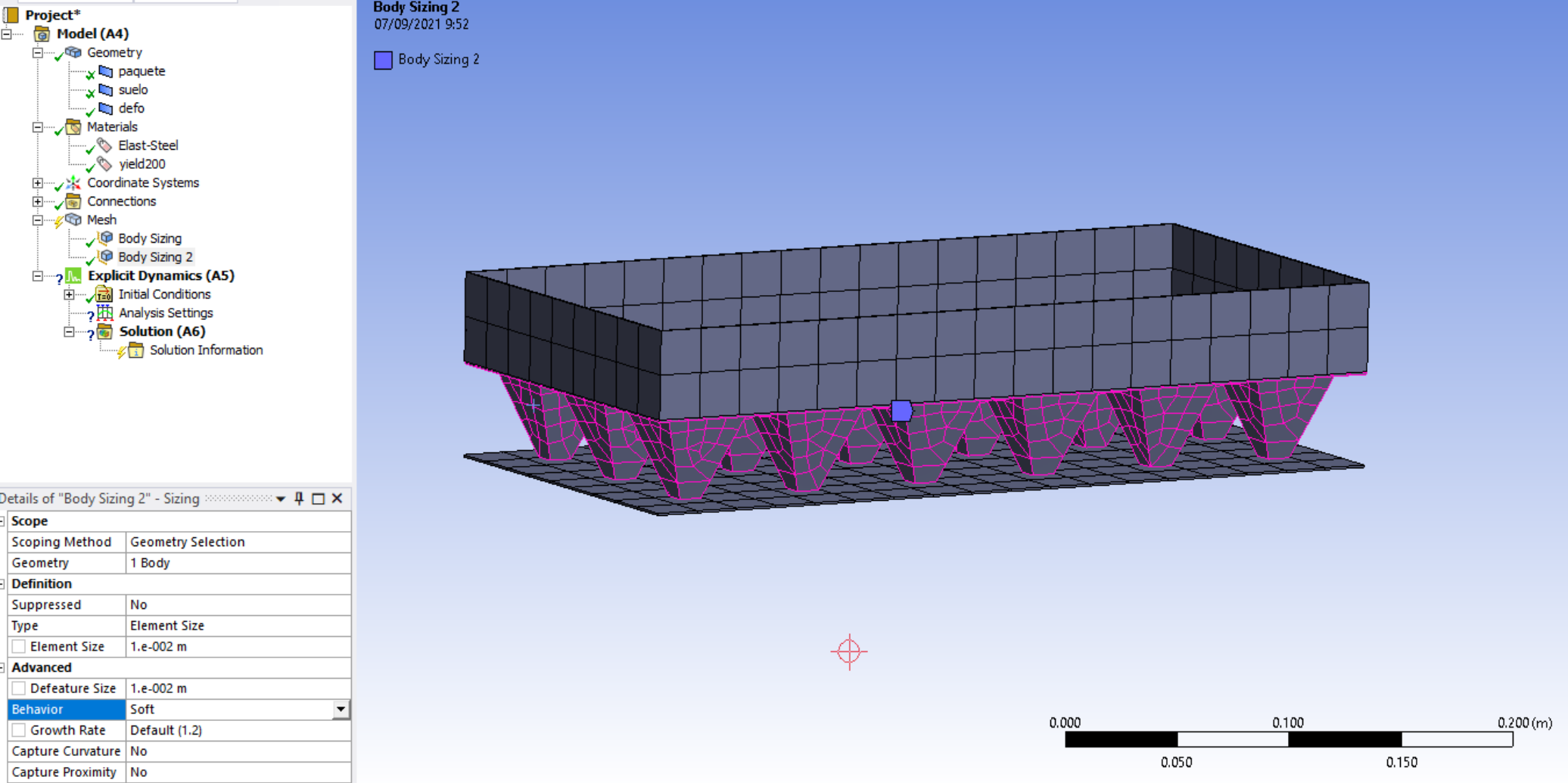


# • Geometría



# • Ponemos espesores y materiales

Para empezar con malla de 10mm.



**Project\***

- Model (A4)
  - Geometry
    - paquete
    - suelo
    - defo
  - Materials
    - Elast-Steel
    - yield200
  - Coordinate Systems
  - Connections
  - Mesh
    - Body Sizing
    - Body Sizing 2
  - Explicit Dynamics (A5)
    - Initial Conditions
    - Analysis Settings
    - Solution (A6)
      - Solution Information

**Body Sizing 2**

07/09/2021 9:52

Body Sizing 2

Details of "Body Sizing 2" - Sizing

Scope

Scoping Method	Geometry Selection
Geometry	1 Body

Definition

Suppressed	No
Type	Element Size
<input type="checkbox"/> Element Size	1.e-002 m

Advanced

<input type="checkbox"/> Defeature Size	1.e-002 m
---	-----------

Behavior	Soft
----------	------

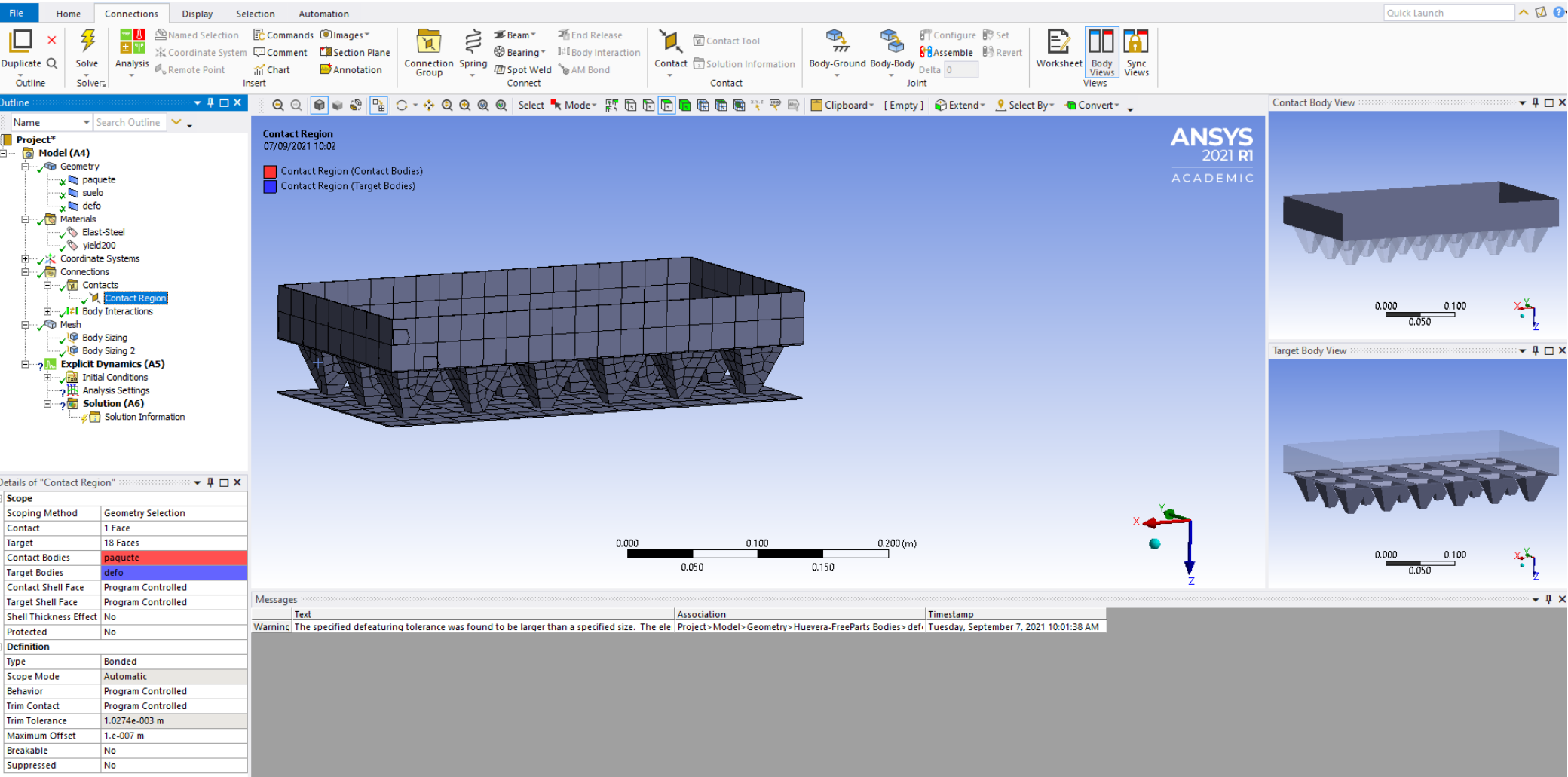
<input type="checkbox"/> Growth Rate	Default (1.2)
--------------------------------------	---------------

Capture Curvature	No
-------------------	----

Capture Proximity	No
-------------------	----

0.000 0.050 0.100 0.150 0.200 (m)

# • Pegar paquete a defo



The screenshot displays the ANSYS 2021 R1 Academic software interface. The main window shows a 3D model of a rectangular block with a grid of supports underneath. The 'Contact Region' is highlighted in red, and the 'Target Bodies' are highlighted in blue. The 'Contact Region' is labeled 'Contact Region (Contact Bodies)' and the 'Target Bodies' are labeled 'Contact Region (Target Bodies)'. The 'Contact Region' is defined by 18 faces of the 'paquete' body, and the 'Target Bodies' are the 'defo' bodies.

The 'Details of "Contact Region"' panel shows the following settings:

Scope	
Scoping Method	Geometry Selection
Contact	1 Face
Target	18 Faces
Contact Bodies	paquete
Target Bodies	defo
Contact Shell Face	Program Controlled
Target Shell Face	Program Controlled
Shell Thickness Effect	No
Protected	No
Definition	
Type	Bonded
Scope Mode	Automatic
Behavior	Program Controlled
Trim Contact	Program Controlled
Trim Tolerance	1.0274e-003 m
Maximum Offset	1.e-007 m
Breakable	No
Suppressed	No

The 'Messages' panel at the bottom shows a warning: "The specified defeaturing tolerance was found to be larger than a specified size. The element Project>Model>Geometry>Huevera-FreeParts Bodies>defo".

# • Contacto inferior

Outline

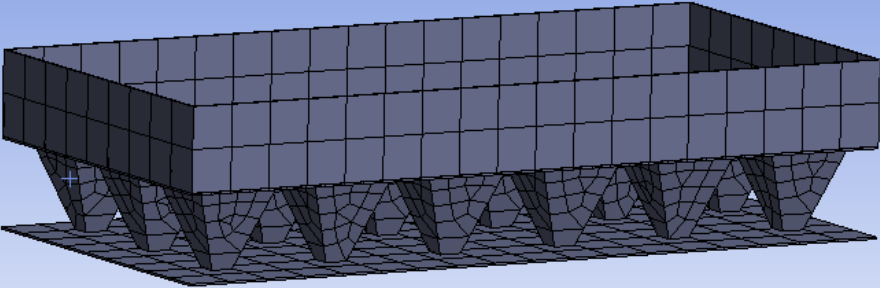
Name Search Outline

Project\*

Model (A4)

- Geometry
  - paquete
  - suelo
  - defo
- Materials
  - Elast-Steel
  - yield200
- Coordinate Systems
- Connections
  - Contacts
    - Contact Region
    - Body Interactions
      - Body Interaction
- Mesh
  - Body Sizing
  - Body Sizing 2
- Explicit Dynamics (A5)
  - Initial Conditions
  - Analysis Settings
  - Solution (A6)
    - Solution Information

Body Interaction  
07/09/2021 10:04



0.000 0.150 0.300 (m)  
0.075 0.225

Details of "Body Interaction"

Scope

Scoping Method	Geometry Selection
Geometry	All Bodies

Definition

Type	Frictional
Friction Coefficient	0.1
Dynamic Coefficient	0.1
Decay Constant	0.
Suppressed	No

Graph Tabular Data

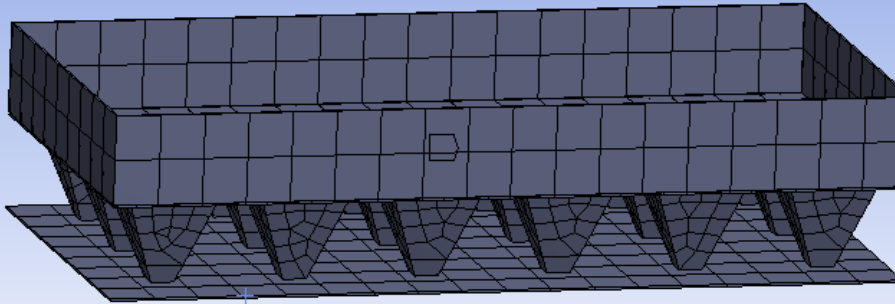
# • Ponemos la masa

**Project\***

- Model (A4)
  - Geometry
    - paquete
    - suelo
    - defo
    - Point Mass 2
  - Materials
    - Elast-Steel
    - yield200
  - Coordinate Systems
  - Connections
    - Contacts
      - Contact Region
  - Body Interactions
    - Body Interaction
  - Mesh
    - Body Sizing
    - Body Sizing 2
- Explicit Dynamics (A5)
  - Initial Conditions
    - Pre-Stress (None)
    - Velocity
  - Analysis Settings
  - Fixed Support
- Solution (A6)
  - Solution Information

**Point Mass 2**  
07/09/2021 10:20

Point Mass 2  
Mass Magnitude: 50. kg  
Location: -0.12;9.e-002;-1.e-003 m



Details of "Point Mass 2" □ ×

Scope	
Scoping Method	Geometry Selection
Applied By	Remote Attachment
Geometry	1 Face
Coordinate System	Global Coordinate System
<input type="checkbox"/> X Coordinate	-0.12 m
<input type="checkbox"/> Y Coordinate	9.e-002 m
<input type="checkbox"/> Z Coordinate	-1.e-003 m
Location	Click to Change
Definition	
<input type="checkbox"/> Mass	50. kg
<input type="checkbox"/> Mass Moment of Inertia X	1. kg·m <sup>2</sup>
<input type="checkbox"/> Mass Moment of Inertia Y	1. kg·m <sup>2</sup>
<input type="checkbox"/> Mass Moment of Inertia Z	1. kg·m <sup>2</sup>
Suppressed	No
Behavior	Rigid
Pinball Region	All

Messages

	Text	Association	Timestamp
Error	Deformable behavior for remote entities is not supported in Explicit Dynamics systems. Pl	Project>Model>Explicit Dynamics>Solution	Tuesday, September 7, 2021 10:20:27 AM
Error	Failed to submit solve job. See Solution Information, which may contain more details.	Project>Model>Explicit Dynamics>Solution	Tuesday, September 7, 2021 10:20:27 AM
Warning	The specified defeaturing tolerance was found to be larger than a specified size. The ele	Project>Model>Geometry>Huevera-FreeParts Bodies>def	Tuesday, September 7, 2021 10:01:38 AM

# • Velocidad inicial

Outline

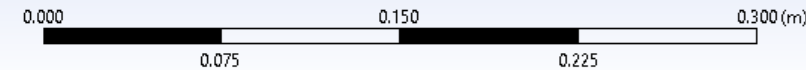
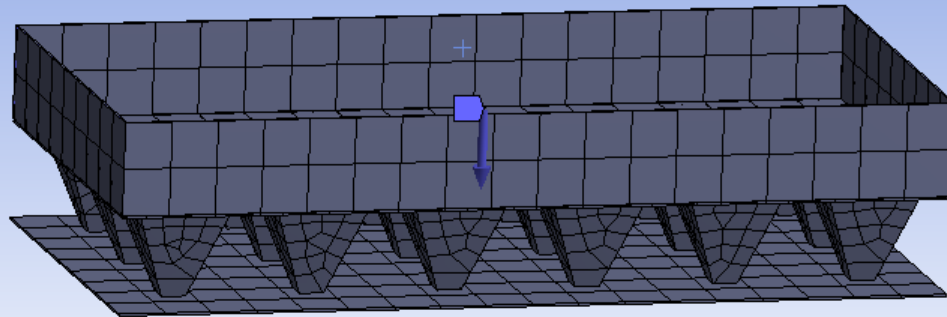
Name Search Outline

**Project\***

- Model (A4)**
  - Geometry
    - paquete
    - suelo
    - defo
    - Point Mass 2
  - Materials
    - Elast-Steel
    - yield200
  - Coordinate Systems
  - Connections
    - Contacts
      - Contact Region
    - Body Interactions
      - Body Interaction
  - Mesh
    - Body Sizing
    - Body Sizing 2
  - Explicit Dynamics (A5)**
    - Initial Conditions
      - Pre-Stress (None)
      - Velocity
    - Analysis Settings
    - Solution (A6)**
      - Solution Information

**A: Explicit Dynamics**  
Velocity  
07/09/2021 10:17

Velocity: 3.5 m/s



Details of "Velocity"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Input Type	Velocity
Pre-Stress Environment	None Available
Define By	Components
Coordinate System	Global Coordinate System
<input type="checkbox"/> X Component	0. m/s
<input type="checkbox"/> Y Component	0. m/s
<input checked="" type="checkbox"/> Z Component	3.5 m/s
Suppressed	No

Messages

Text	Association	Timestamp
Warning: The specified defeaturing tolerance was found to be larger than a specified size. The ele	Project> Model> Geometry> Huevera-FreeParts Bodies> defo	Tuesday, September 7, 2021 10:01:38 AM

# • Fijamos contorno

Outline

Name Search Outline

**Project\***

- Model (A4)
  - Geometry
    - paquete
    - suelo
    - defo
    - Point Mass 2
  - Materials
    - Elast-Steel
    - yield200
  - Coordinate Systems
  - Connections
    - Contacts
      - Contact Region
    - Body Interactions
      - Body Interaction
  - Mesh
    - Body Sizing
    - Body Sizing 2
  - Explicit Dynamics (A5)**
    - Initial Conditions
      - Pre-Stress (None)
      - Velocity
    - Analysis Settings
    - Fixed Support**
  - Solution (A6)**
    - Solution Information

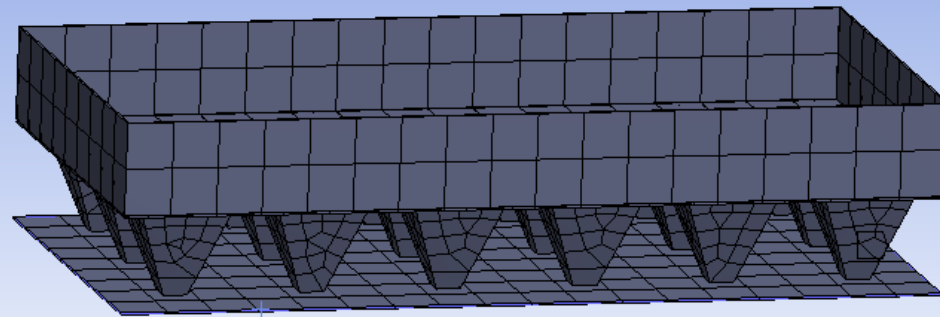
## A: Explicit Dynamics

Fixed Support

Time: 1. s

07/09/2021 10:18

Fixed Support



Graph

Tabular Data

# • Tiempo de simulación

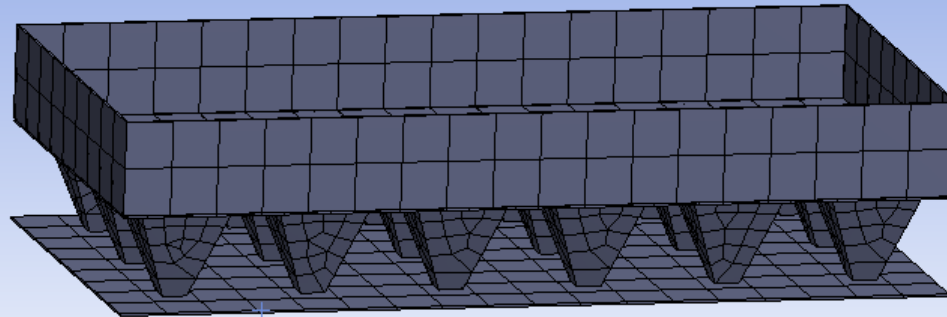
Outline

Name Search Outline

**Project\***

- Model (A4)**
  - Geometry
    - paquete
    - suelo
    - defo
    - Point Mass 2
  - Materials
    - Elast-Steel
    - yield200
  - Coordinate Systems
  - Connections
    - Contacts
      - Contact Region
    - Body Interactions
      - Body Interaction
  - Mesh
    - Body Sizing
    - Body Sizing 2
  - Explicit Dynamics (A5)**
    - Initial Conditions
      - Pre-Stress (None)
      - Velocity
    - Analysis Settings
    - Fixed Support
  - Solution (A6)**
    - Solution Information

**A: Explicit Dynamics**  
Analysis Settings  
Time: 1. s  
07/09/2021 10:19



0.000 0.150 0.300 (m)  
0.075 0.225

Details of "Analysis Settings"

Analysis Settings Preference	
Type	Program Controll...
<b>Step Controls</b>	
Number Of Steps	1
Current Step Number	1
Load Step Type	Explicit Time Inte...
End Time	0.02



Tabular Data

	End Time [s]
1	0.
*	



## • Cálculo inicial

Vemos que el time step no es de  $1.5e-6$  como hasta ahora. Ahora tenemos algún elemento más pequeño en el defo que hace que trabajemos con un time step de  $6.1e-7$  que hace que tengamos un tiempo de cálculo aproximadamente  $\times 2.5$

```
Cycle: 32161, Time: 2.000E-02s, Time Inc.: 6.880E-07s, Progress: 100.00%, Est. Clock Time Remaining: 0s  
Cycle: 32162, Time: 2.000E-02s, Time Inc.: 6.152E-07s, Progress: 100.00%, Est. Clock Time Remaining: -
```

### SIMULATION ELAPSED TIME SUMMARY

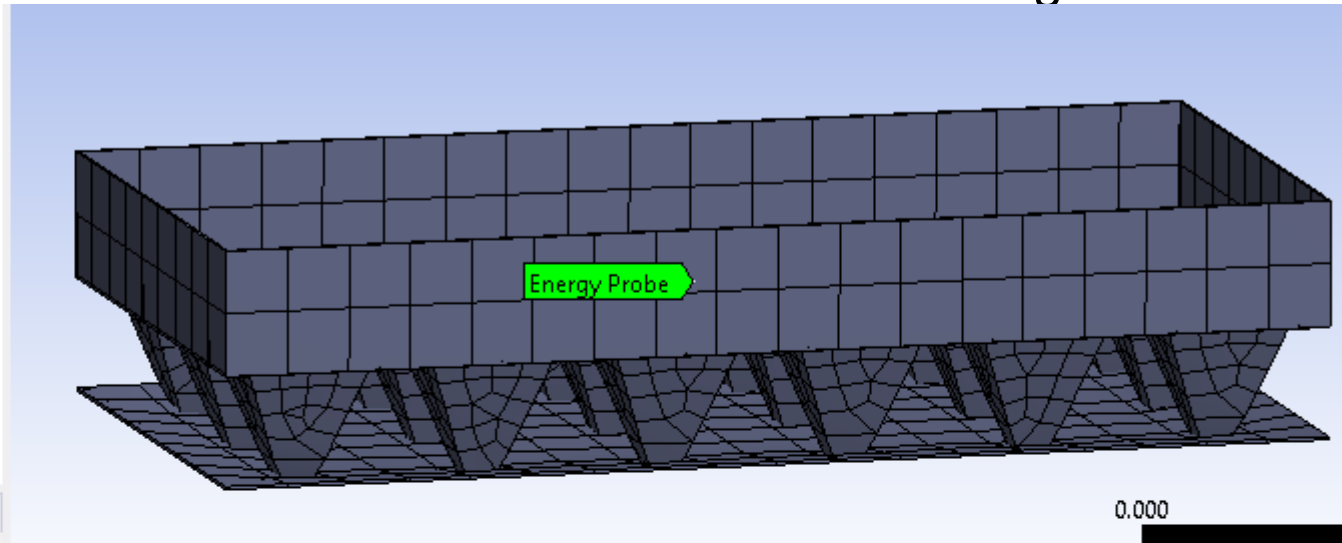
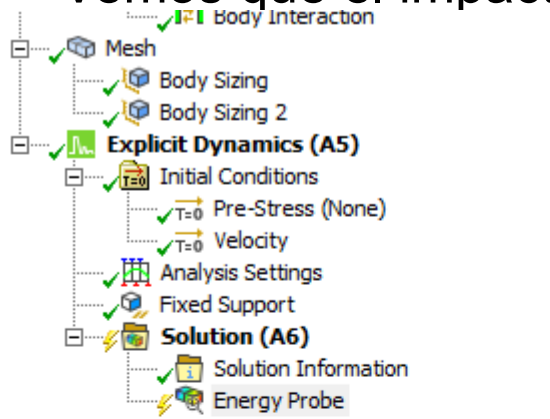
```
EXECUTION FROM CYCLE      0 TO      32162  
ELAPSED RUN TIME IN SOLVER =      1.97292E+00 Minutes  
TOTAL ELAPSED RUN TIME   =      2.04545E+00 Minutes  
JOB RAN OVER      2 WORKERS  
JOB RAN USING Intel MPI  
JOB RAN USING DECOMPOSITION AUTO
```

```
Problem terminated .... wrapup time reached
```

```
*****
```

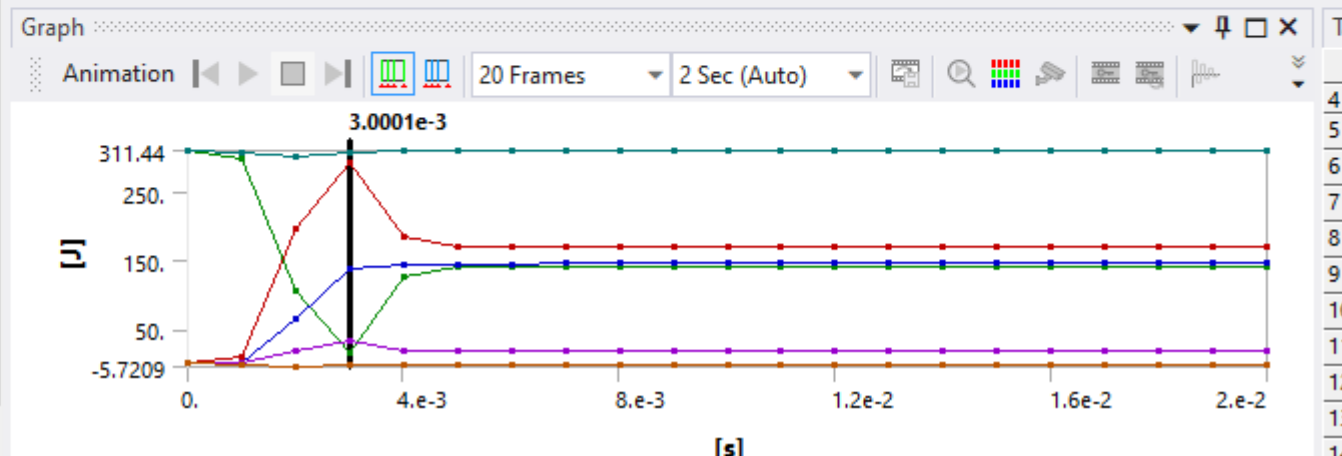
# • Energías

Vemos que el impacto ha sido casi elástico retornando 50% energía.



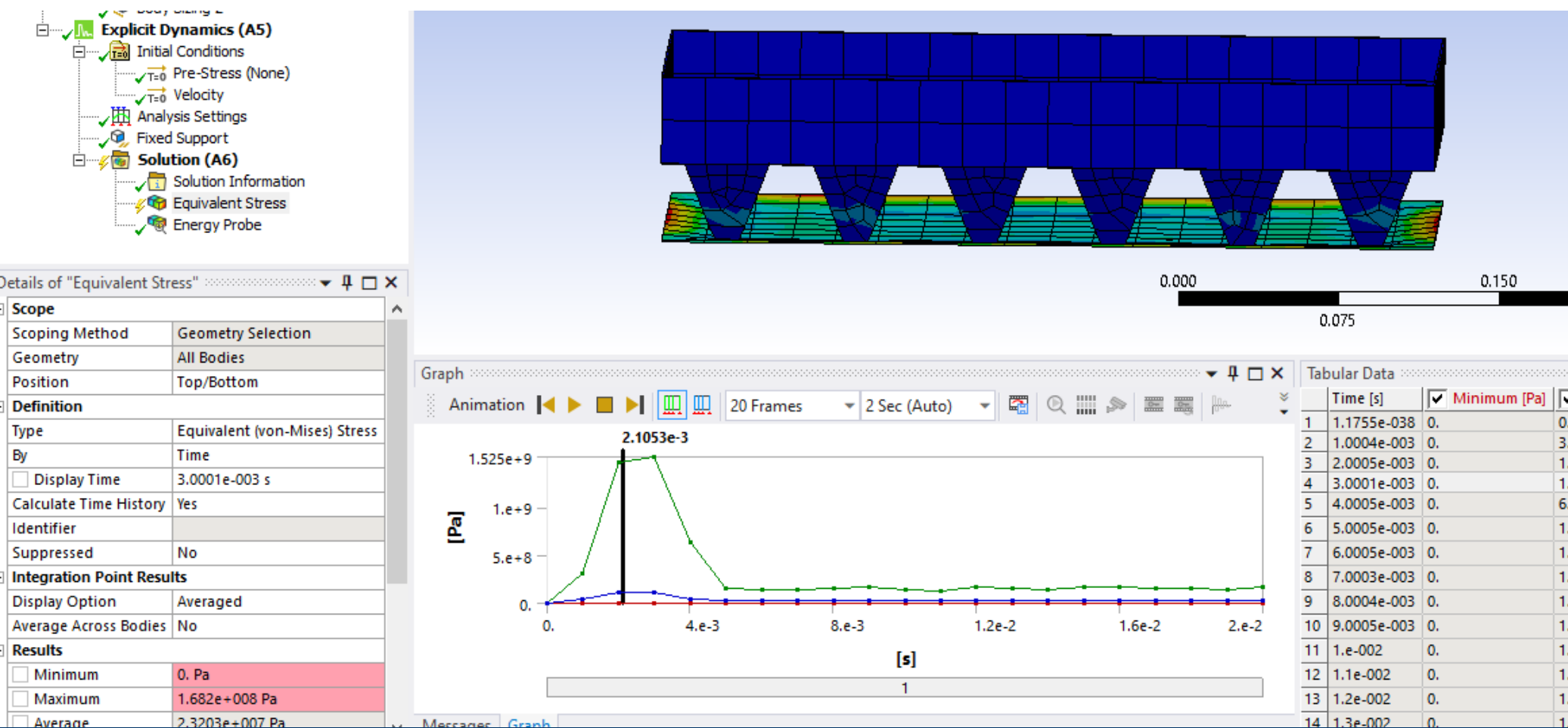
Properties of "Energy Probe"

Definition	
Type	Energy
Geometry	3 Bodies
Compressed	No
Options	
Result Selection	All
Display Time	3.0001e-003 s
Results	
Maximum Value Over Time	
Internal	292.41 J
Kinetic	311.44 J
Plastic Work	145.58 J
Hourglass	31.631 J
Contact	0. J
Total	311.44 J



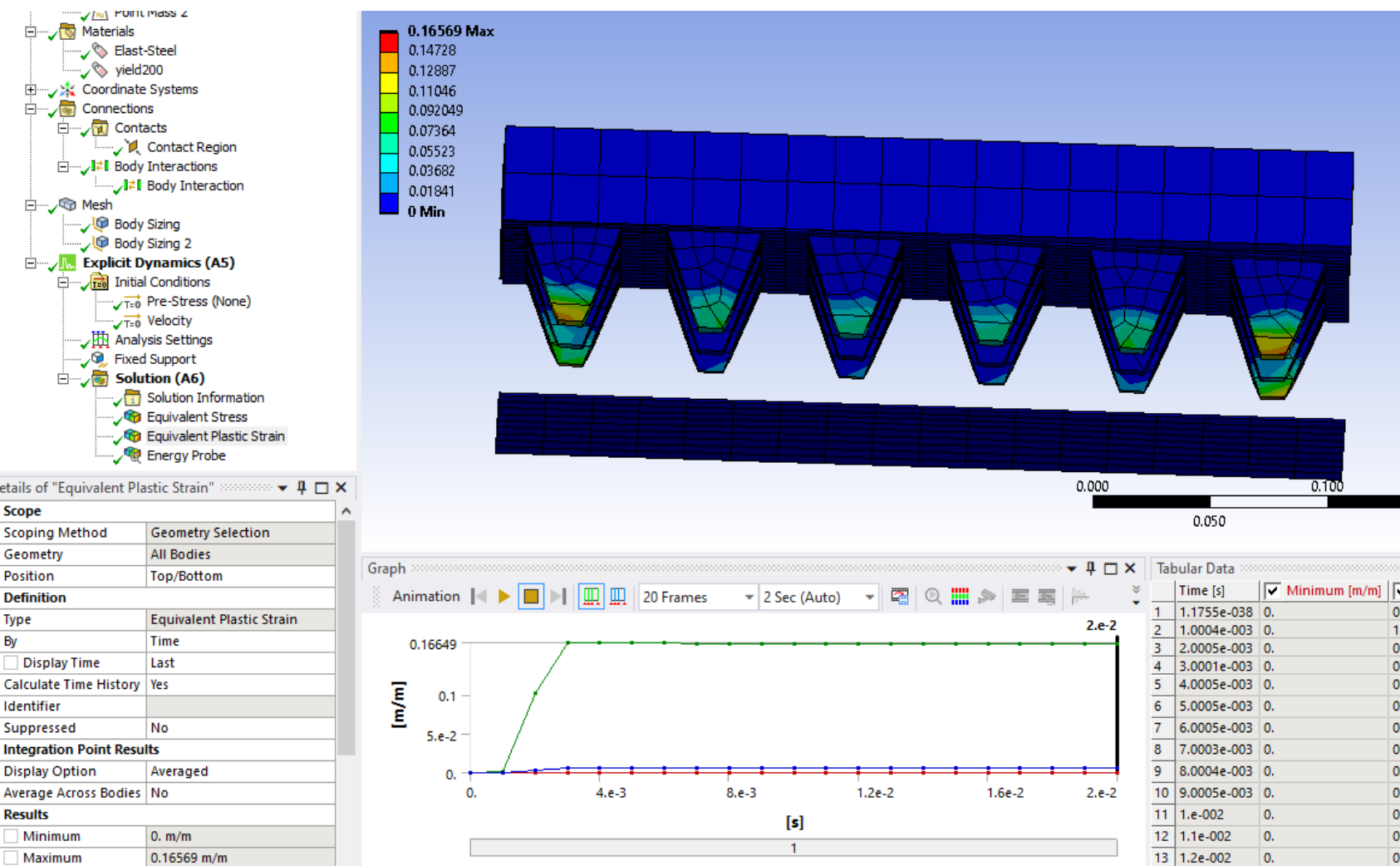
# • Tensiones

Con esta configuración la base elástica es la que tiene la tensión alta



# • Deformación plástica

El defo tiene algo de deformación plástica pero no ha hecho su trabajo



# • Reasignación de propiedades

Primero vamos a poner la pared de abajo toda restringida y no sólo en el contorno. Luego para el defo vamos a hacer un material con propiedades divididas por 100 de módulo Young y Yield para que deforme más y no estropee el time step.

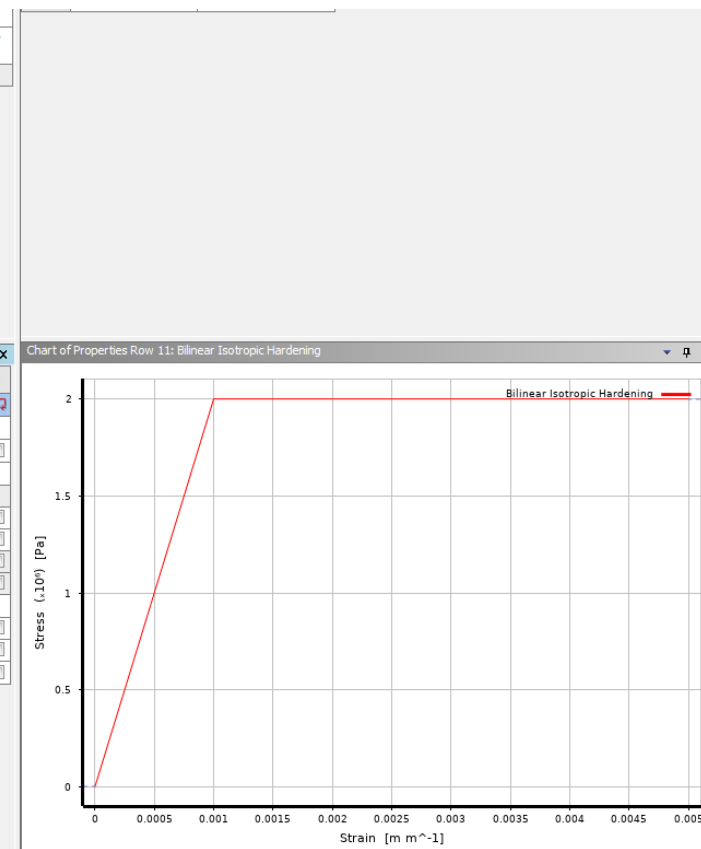
2, Table 5-110.1

Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1

Click here to add a new material

Properties of Outline Row 4: yield200-pulp

	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Density	2000	kg m <sup>-3</sup>		
4	Isotropic Elasticity				
5	Derive from	Young's Modulus and Poisson...			
6	Young's Modulus	2E+09	Pa		
7	Poisson's Ratio	0.3			
8	Bulk Modulus	1.6667E+09	Pa		
9	Shear Modulus	7.6923E+08	Pa		
10	Bilinear Isotropic Hardening				
11	Yield Strength	2E+06	Pa		
12	Tangent Modulus	0	Pa		
13	Specific Heat Constant Pressure, C <sub>p</sub>	434	J kg <sup>-1</sup> C <sup>-1</sup>		



# • Resultados

Ahora puede ir con un time step grande y corre rápido.

```
Cycle:      7374, Time: 1.623E-02s, Time Inc.: 1.889E-06s, Progress: 81.15%, Est. Clock Time Remaining: 9s
```

## SIMULATION ELAPSED TIME SUMMARY

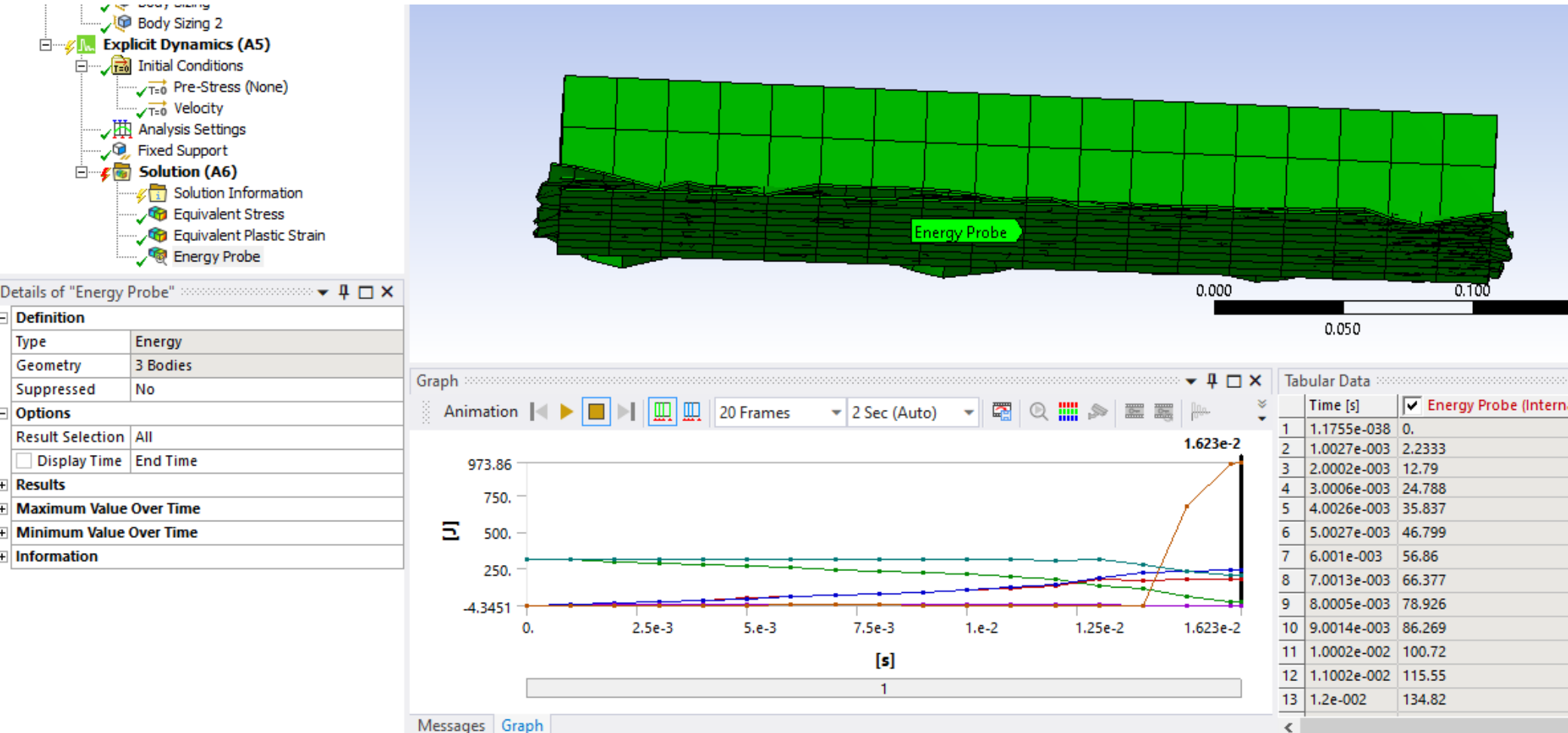
```
EXECUTION FROM CYCLE      0 TO      7374
ELAPSED RUN TIME IN SOLVER =      6.50100E-01 Minutes
TOTAL ELAPSED RUN TIME    =      6.97833E-01 Minutes
JOB RAN OVER      2 WORKERS
JOB RAN USING Intel MPI
JOB RAN USING DECOMPOSITION AUTO
```

```
Problem terminated .... energy error too large
Problem terminated .... energy error too large
```

```
*****
```

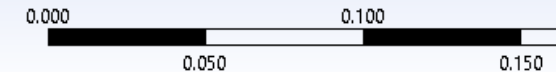
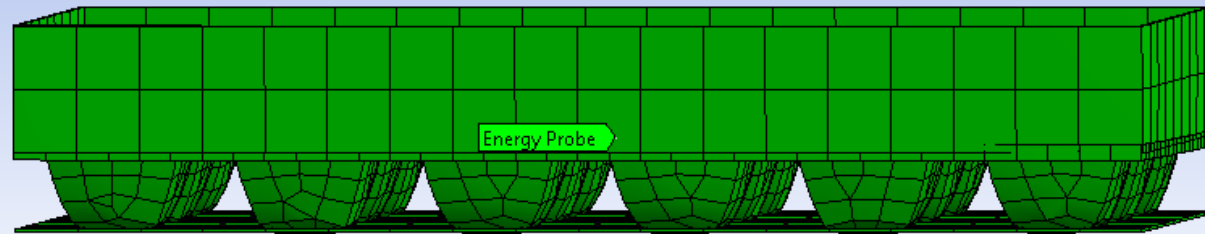
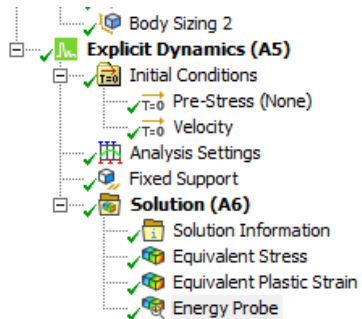
# • Energías

Como se puede ver ahora es demasiado débil con espesor 1[mm] y por tanto no frena a tiempo colapsando el defo



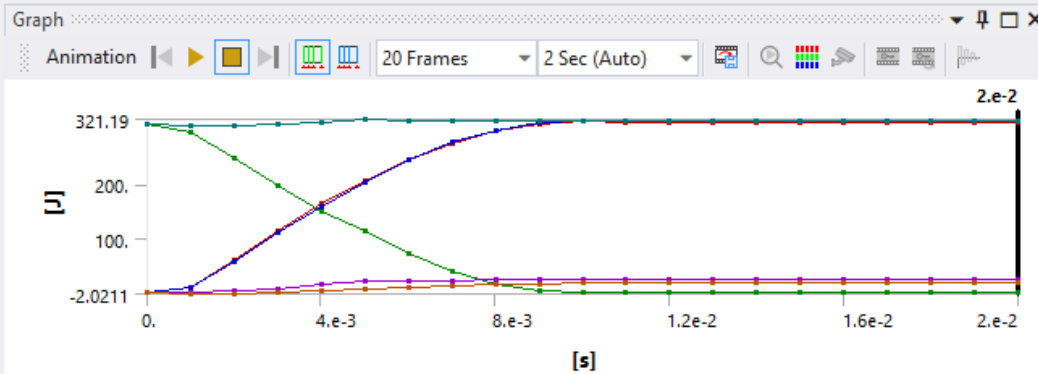
# • Juego con espesores

Repetimos el cálculo jugando con espesores hasta un máximo que podamos fabricar. Con 5[mm] vemos que ya paramos la caja pero usando poco espacio y por tanto mucha fuerza y aceleración.



Details of "Energy Probe"

Definition	
Type	Energy
Geometry	3 Bodies
Suppressed	No
Options	
Result Selection	All
<input type="checkbox"/> Display Time	End Time
Results	
<input type="checkbox"/> Maximum Value Over Time	
<input type="checkbox"/> Minimum Value Over Time	
<input type="checkbox"/> Information	



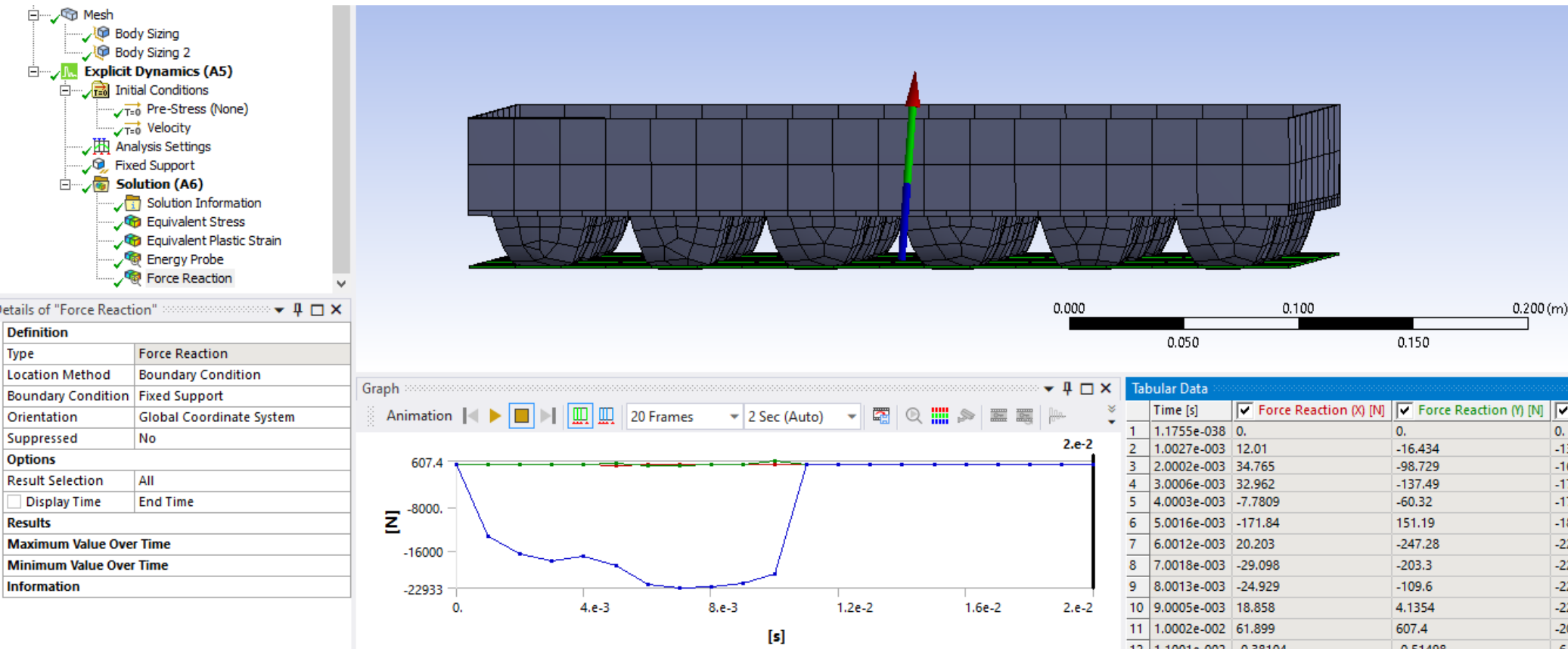
Tabular Data

Time [s]	Energy Probe (Internal) [J]	Energy	
1	1.1755e-038	0.	311.44
2	1.0027e-003	10.557	298.41
3	2.0002e-003	60.251	248.78
4	3.0006e-003	114.02	197.64
5	4.0003e-003	164.94	151.58
6	5.0016e-003	207.37	113.82
7	6.0012e-003	245.54	73.734
8	7.0018e-003	277.07	40.336
9	8.0013e-003	300.19	16.852
10	9.0005e-003	313.43	3.6282
11	1.0002e-002	317.09	5.7342e-00
12	1.1001e-002	316.59	0.66987



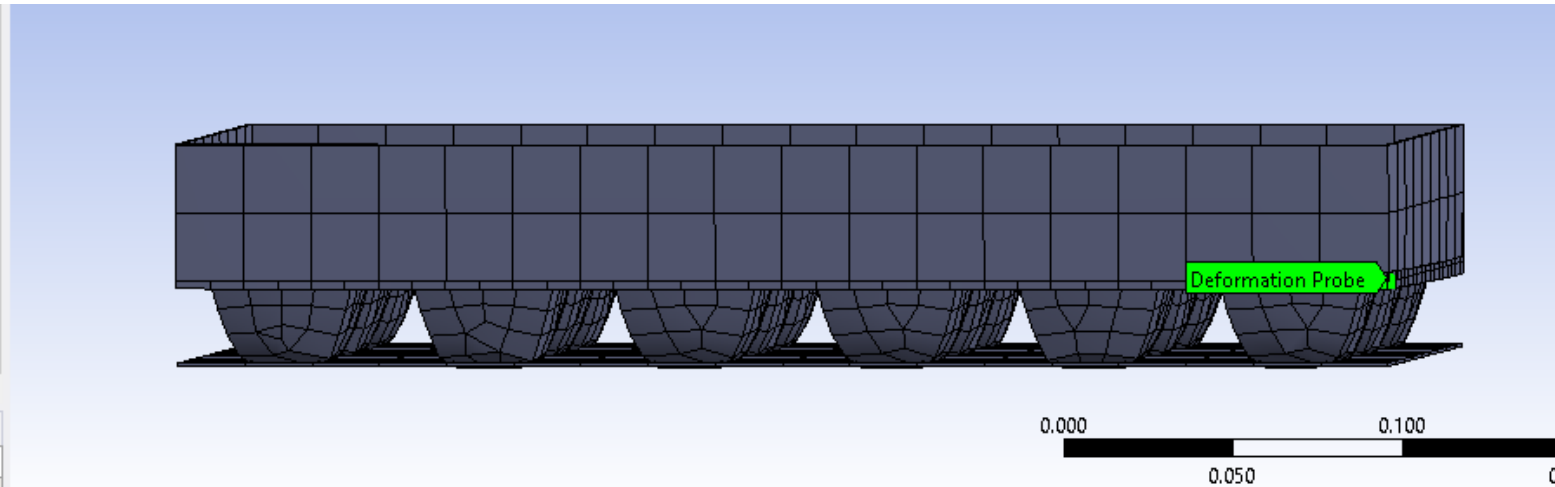
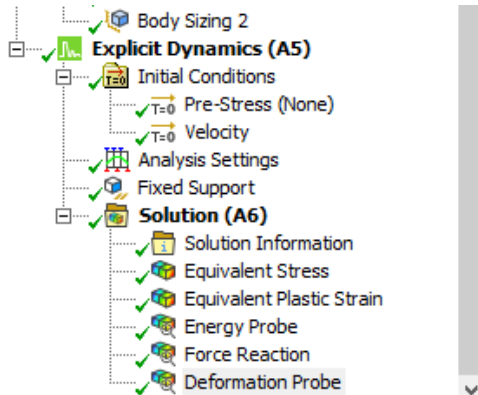
# • Fuerza reacción

Buscábamos una fuerza de reacción de unos 15[kN] y tenemos unos 23[kN] por lo que usamos menos espacio



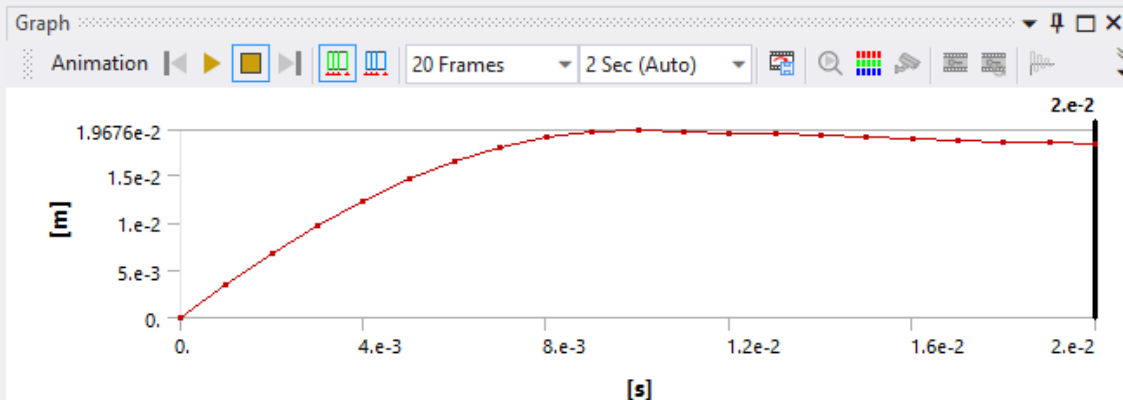
# • Espacio utilizado

Buscábamos usar unos 20[mm] para llegar a usar más si es más débil sin problema.



Details of "Deformation Probe"

Definition	
Type	Deformation
Location Method	Geometry Selection
Geometry	1 Vertex
Orientation	Global Coordinate System
Suppressed	No
Options	
Result Selection	Z Axis
<input type="checkbox"/> Display Time	End Time
Spatial Resolution	Use Maximum
Results	
Maximum Value Over Time	
Minimum Value Over Time	
Information	

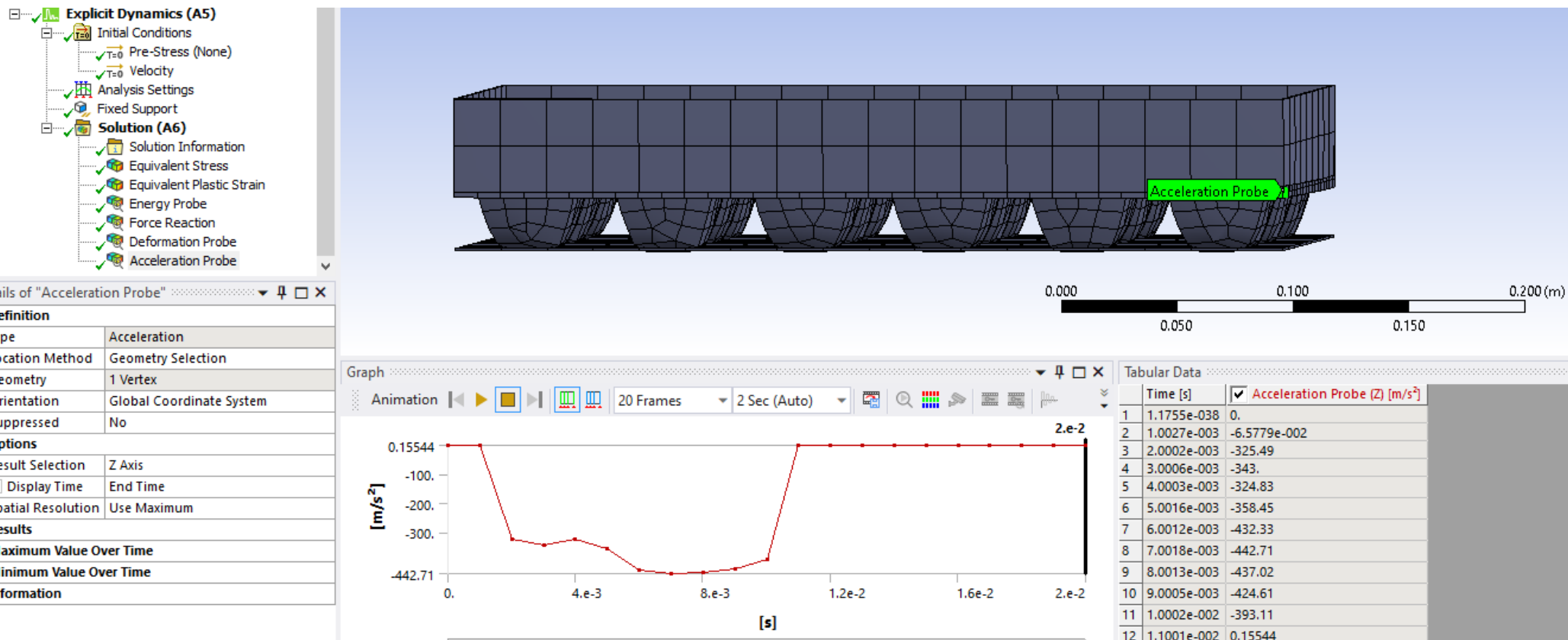


Tabular Data

Time [s]	Deformation Probe (Z) [m]
1	1.1755e-038 0.
2	1.0027e-003 3.4487e-003
3	2.0002e-003 6.6942e-003
4	3.0006e-003 9.6273e-003
5	4.0003e-003 1.2217e-002
6	5.0016e-003 1.4486e-002
7	6.0012e-003 1.6391e-002
8	7.0018e-003 1.7868e-002
9	8.0013e-003 1.8903e-002
10	9.0005e-003 1.9501e-002
11	1.0002e-002 1.9676e-002

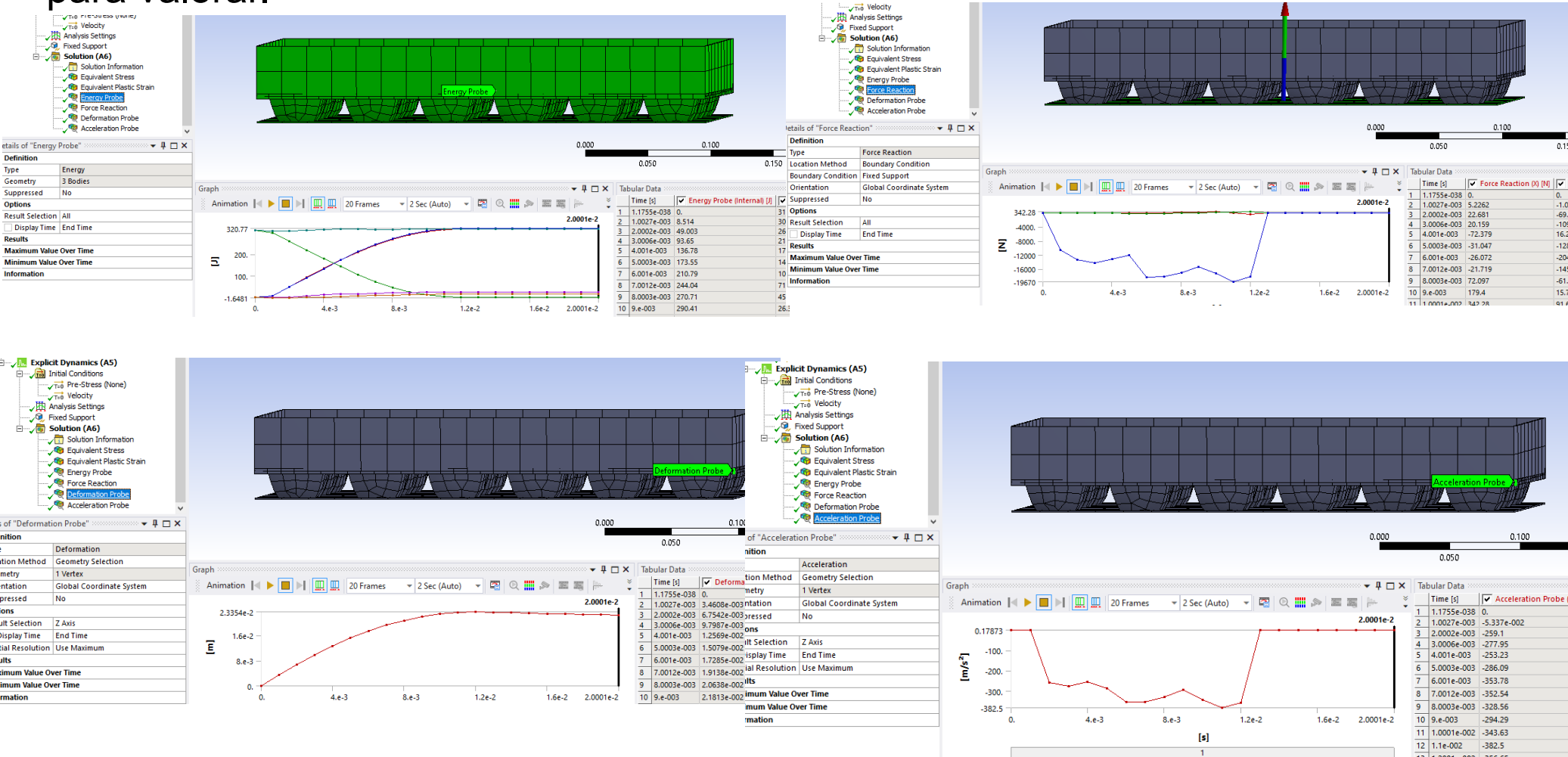
# • Aceleración

Todo el diseño se basa en tener una aceleración constante de unos 31[g] pero la obtenida se parece a un impacto elástico al principio y luego está en torno a los 44[g].



## • Cambio a espesor 4[mm]

Haremos una tabla de espesores y valores de fuerza, desplazamiento y aceleración para valorar.



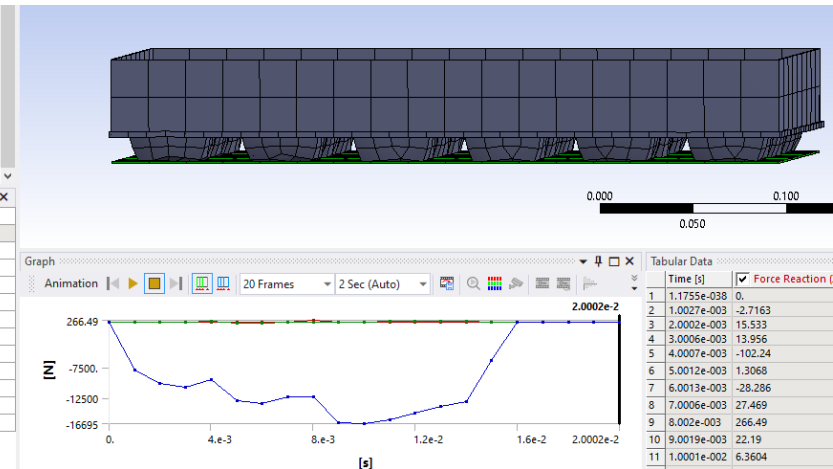
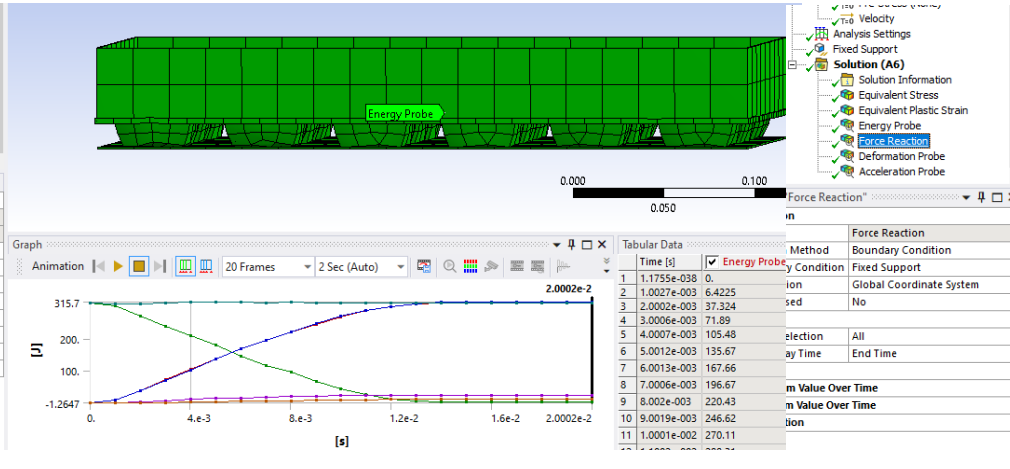
## • Cambio a espesor 3[mm]

Repetimos

Velocity  
Analysis Settings  
Fixed Support  
**Solution (A6)**  
Solution Information  
Equivalent Stress  
Equivalent Plastic Strain  
Energy Probe  
Force Reaction  
Deformation Probe  
Acceleration Probe

Energy Probe  
Energy  
3 Bodies  
No

Selection All  
End Time  
Maximum Value Over Time  
Minimum Value Over Time

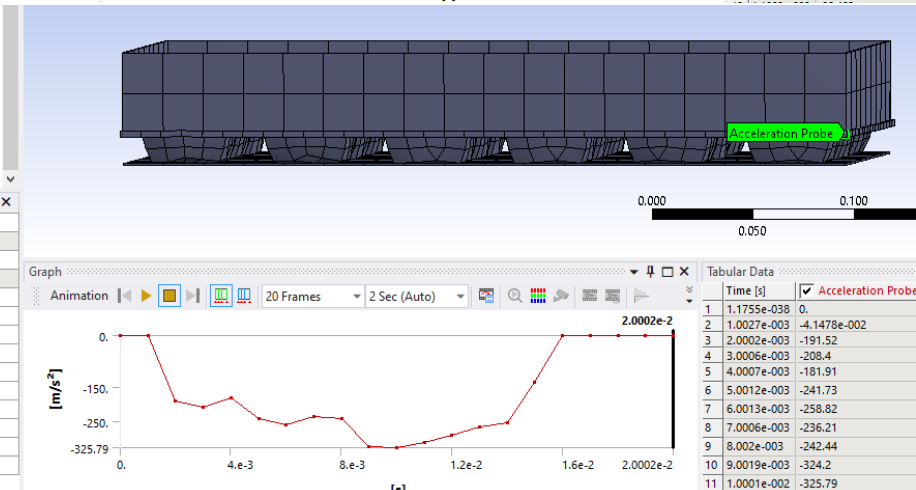
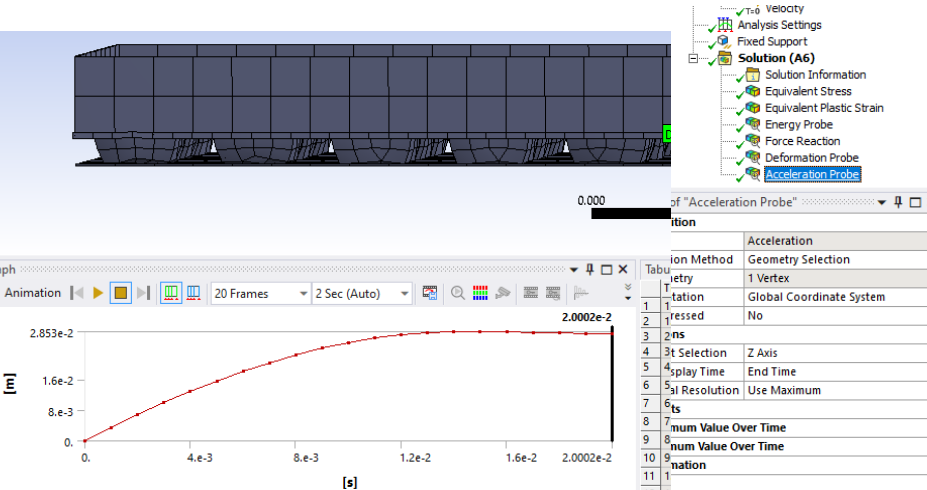


Fixed Support  
**Solution (A6)**  
Solution Information  
Equivalent Stress  
Equivalent Plastic Strain  
Energy Probe  
Force Reaction  
Deformation Probe  
Acceleration Probe

Acceleration Probe  
Acceleration  
1 Vertex  
Global Coordinate System  
No

Z Axis  
End Time  
Use Maximum

Maximum Value Over Time  
Minimum Value Over Time



## • Cambio a espesor 2[mm]

Details of "Energy Probe"

Time [s]	Energy [J]
0	313.54
0.004	200
0.008	100
0.012	-0.85726

Details of "Force Reaction"

Time [s]	Force Reaction [N]
0	99.782
0.004	-7500
0.008	-12500
0.012	-15611
0.016	-12500
0.02	99.782

Details of "Deformation Probe"

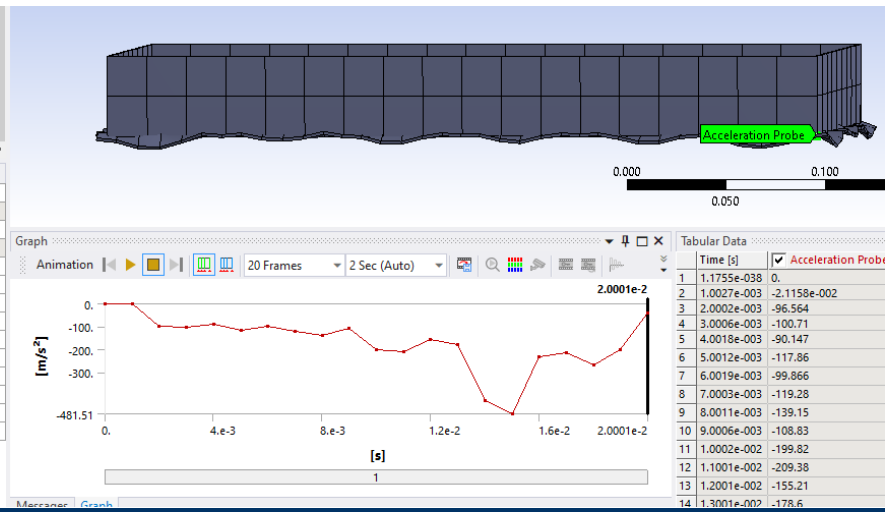
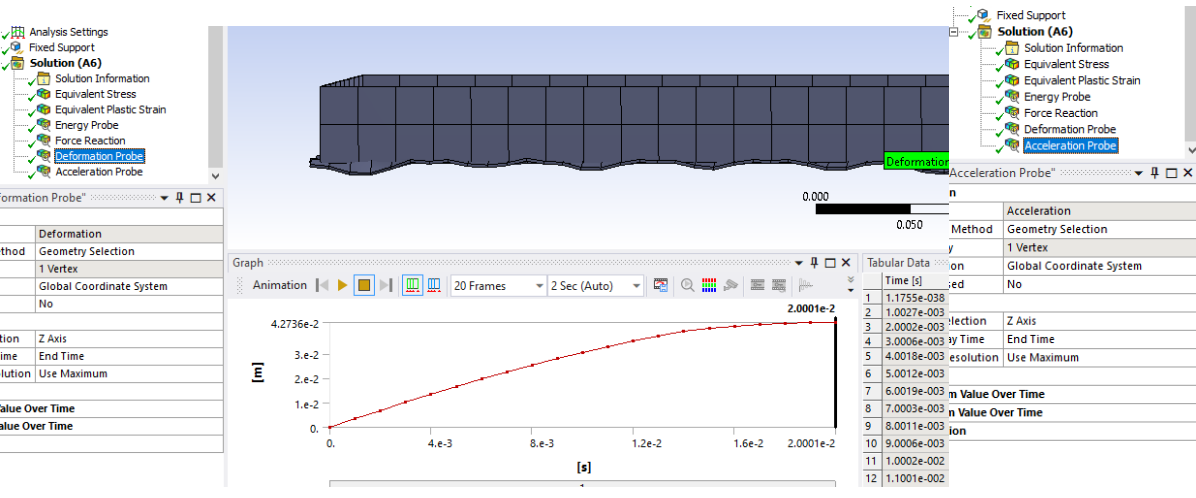
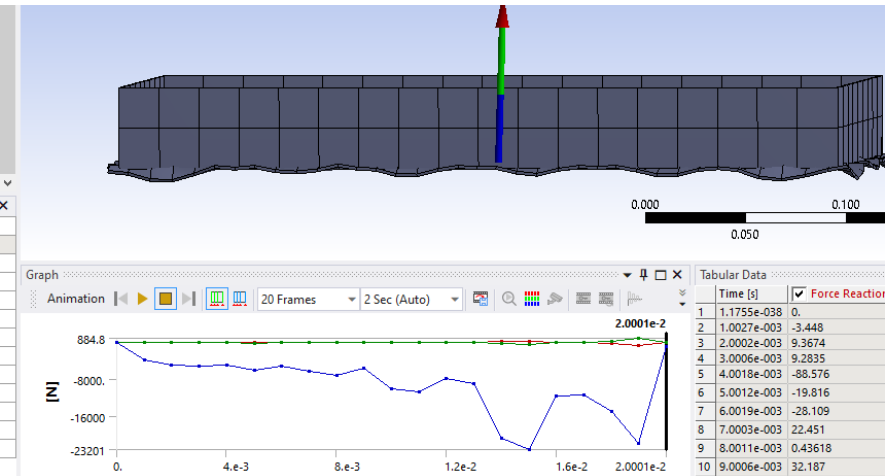
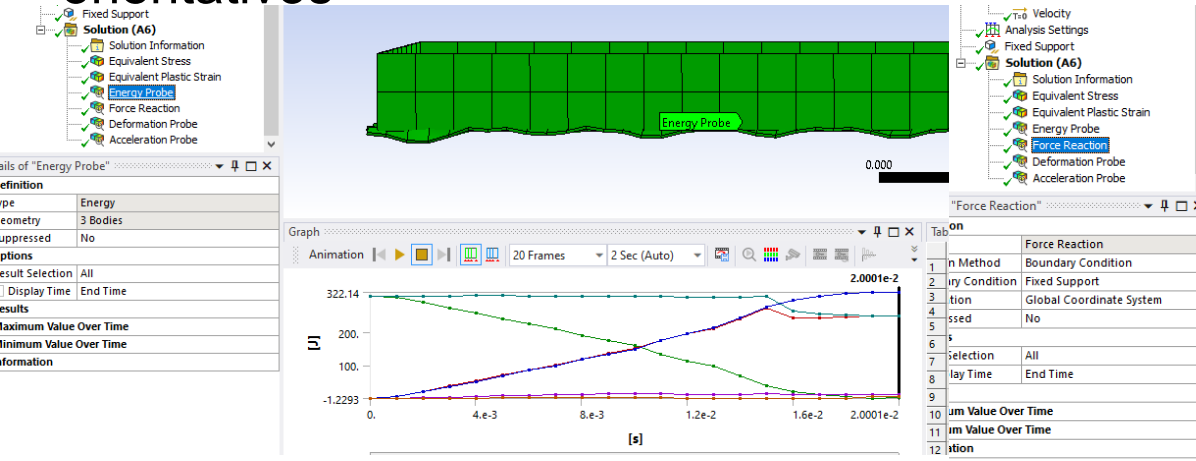
Time [s]	Deformation [m]
0	0
0.004	1e-2
0.008	2e-2
0.012	3e-2
0.016	3.5e-2
0.02	3.7346e-2

Details of "Acceleration Probe"

Time [s]	Acceleration [m/s^2]
0	0
0.004	-150
0.008	-250
0.012	-307.15
0.016	-200
0.02	0

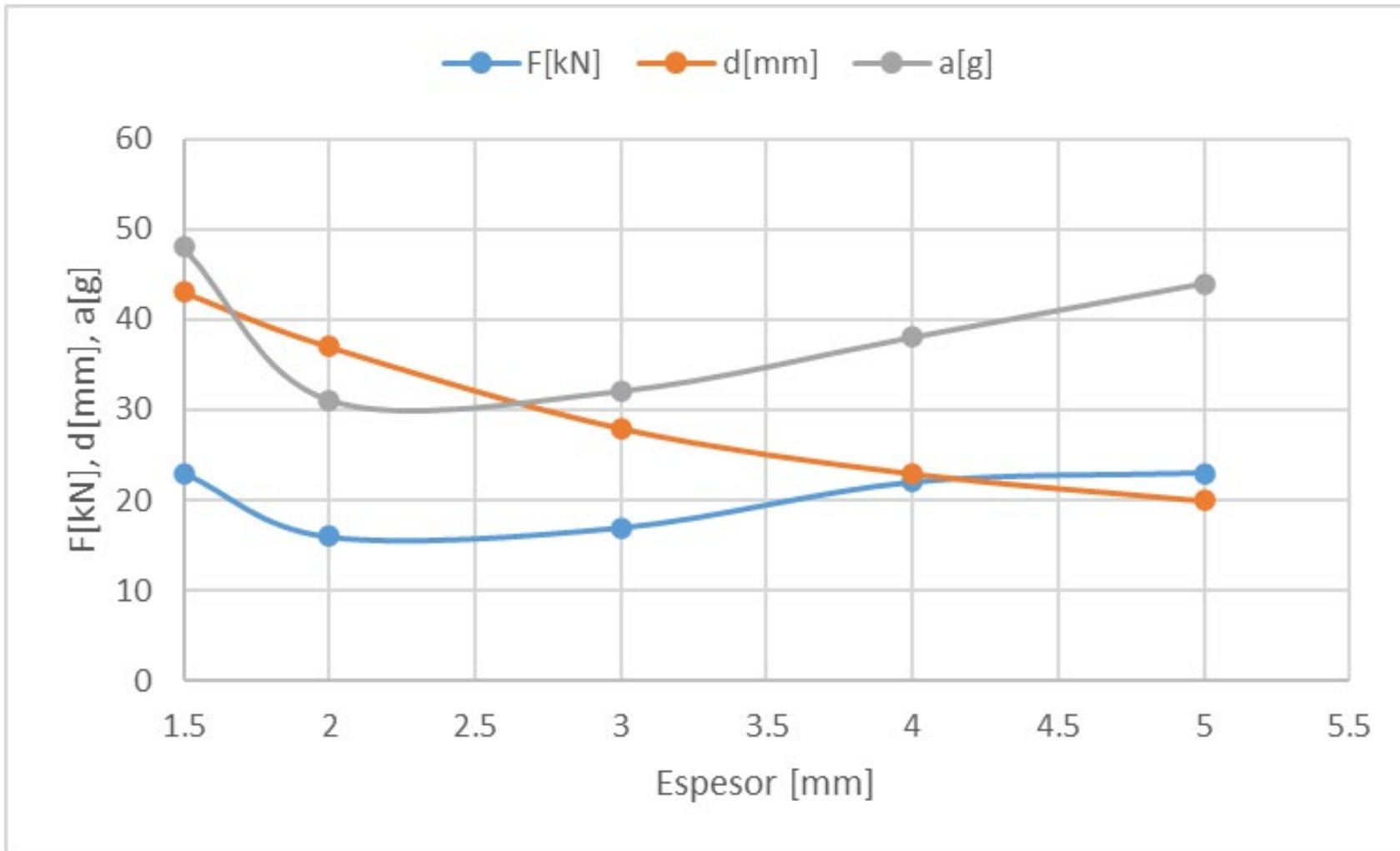
## • Cambio espesor 1.5[mm]

Este es un valor donde se impacta la base y se disparan fuerzas. Valores sólo orientativos



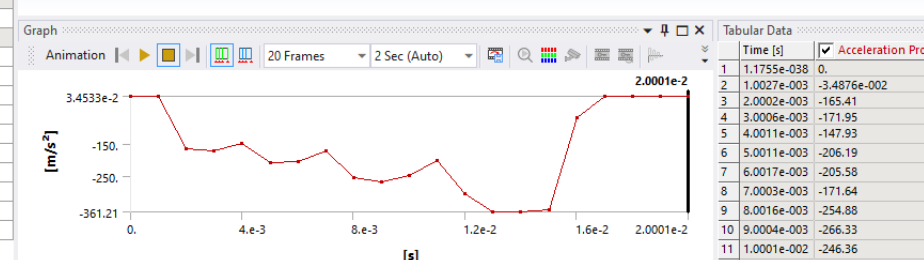
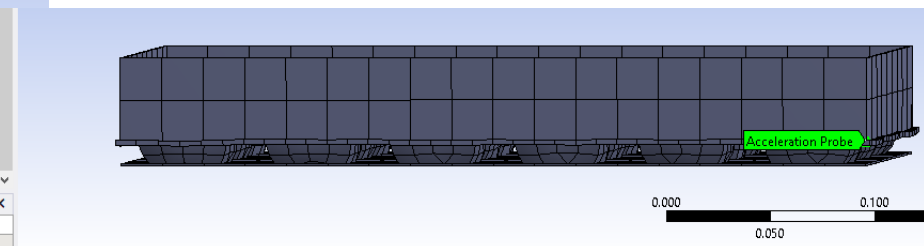
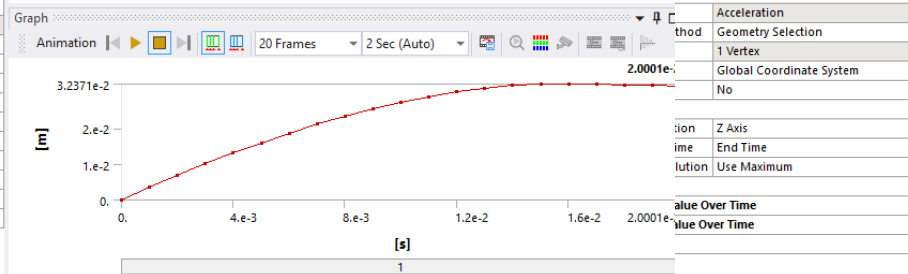
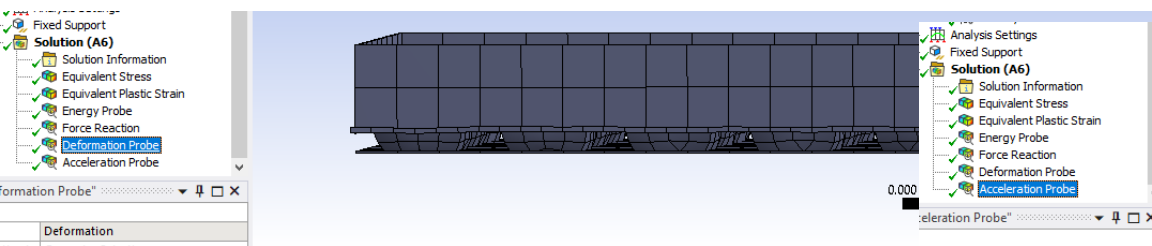
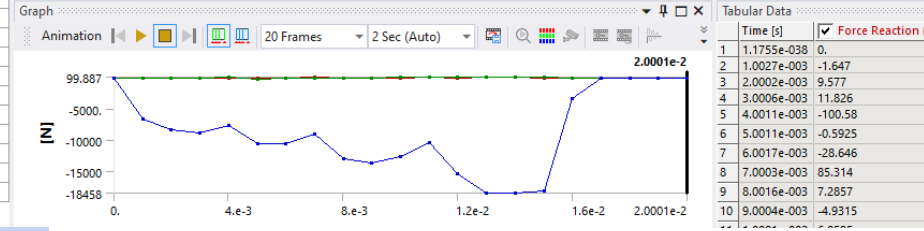
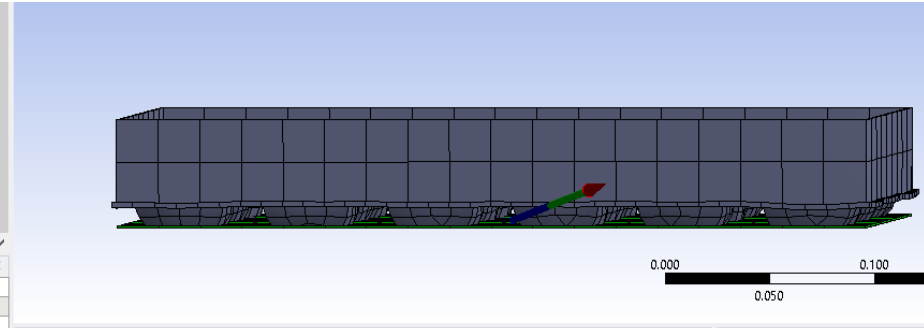
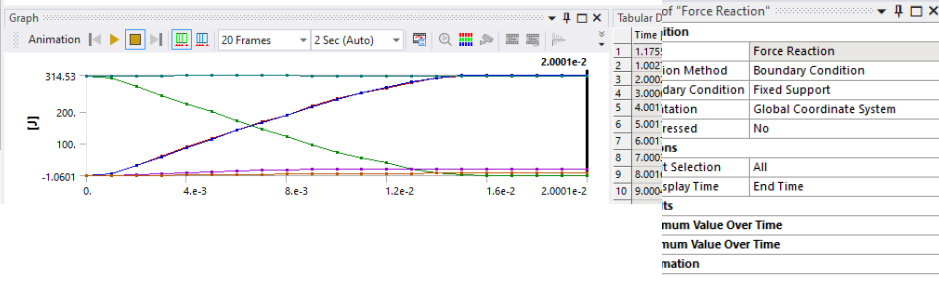
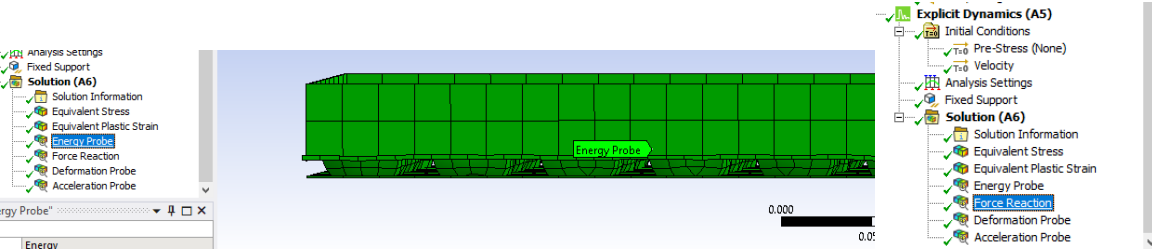
## • Análisis

Si plotemaos vemos el óptimo puede estar entre 2 y 3[mm]



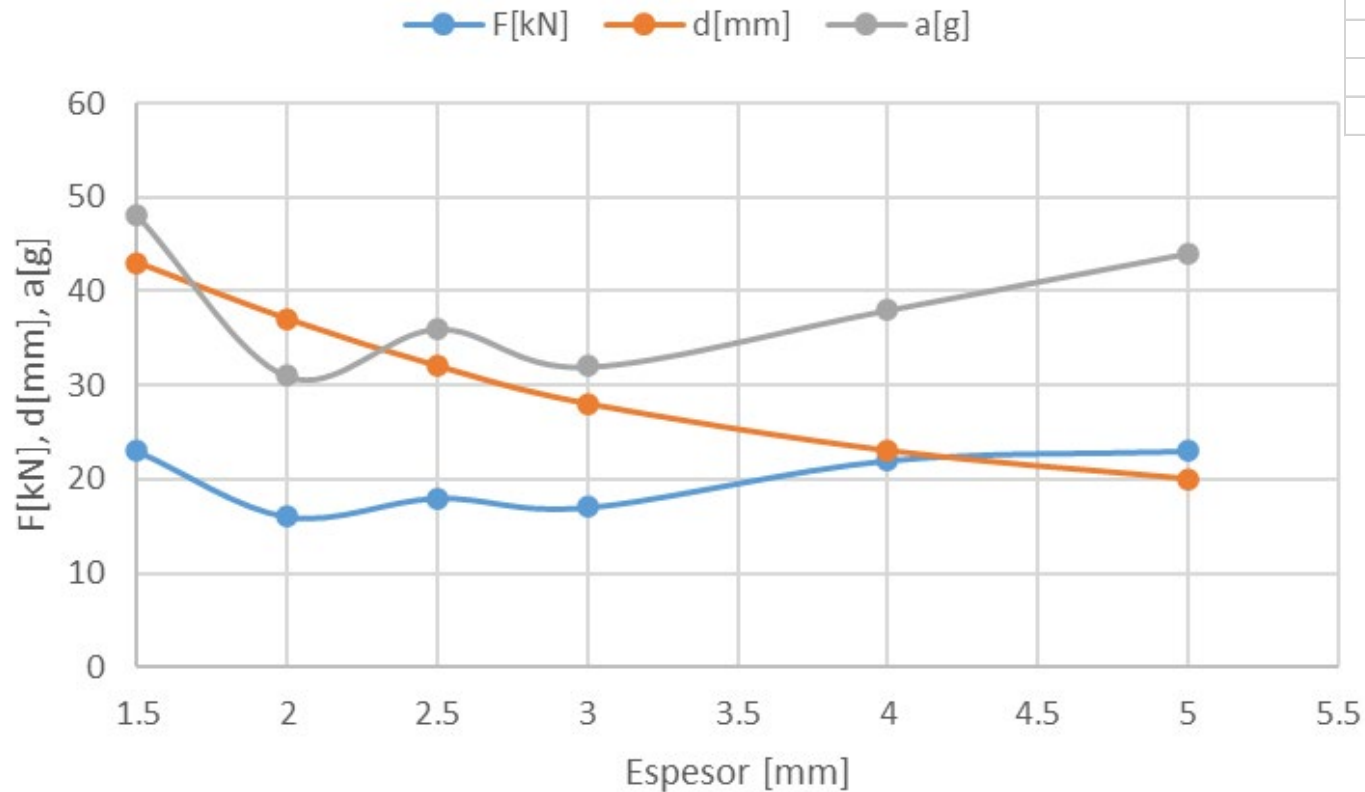


## • Repetimos para 2.5[mm]



## • Análisis

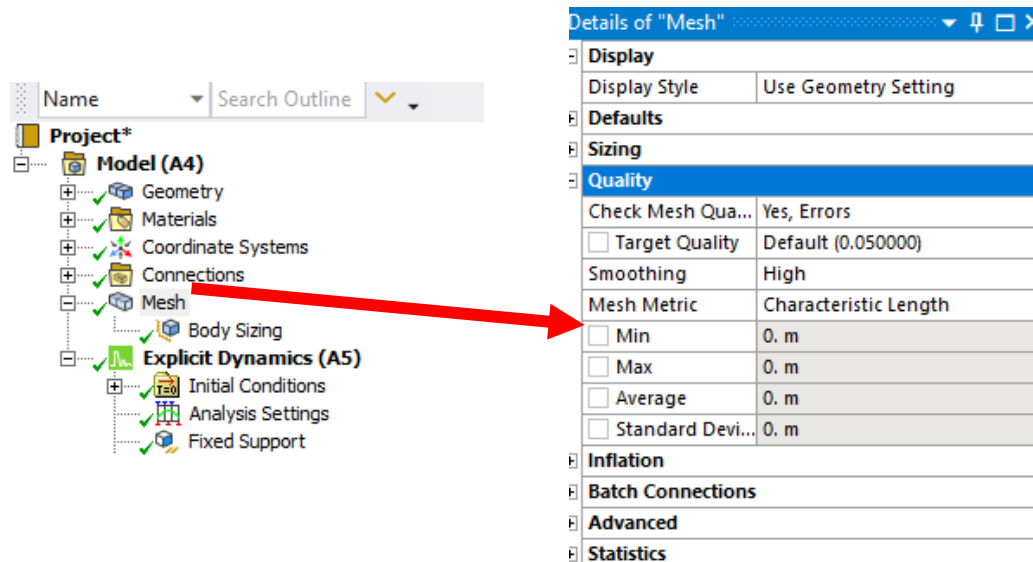
Si diseñamos con un espesor de 2.5[mm] puede que tengamos valores entre 2 y 3[mm]



espesor mm	F kN	d mm	a g
1.5	23	43	48
2	16	37	31
2.5	18	32	36
3	17	28	32
4	22	23	38
5	23	20	44

## • Modelo sencillo

Para entender el modelo podemos colapsar una única huevera-defo de las 18 que hay. Para ello pondremos  $50/18=2.78[\text{kg}]$  para ver si nos da una fuerza de  $15/18=0.833[\text{kN}]$



The image shows a screenshot of the ANSYS software interface. On the left is the 'Project\*' tree view, and on the right is the 'Details of "Mesh"' dialog box.

**Project\* Tree View:**

- Project\*
  - Model (A4)
    - Geometry
    - Materials
    - Coordinate Systems
    - Connections
    - Mesh
    - Body Sizing
  - Explicit Dynamics (A5)
    - Initial Conditions
    - Analysis Settings
    - Fixed Support

**Details of "Mesh" Dialog Box:**

Details of "Mesh"	
<b>Display</b>	
Display Style	Use Geometry Setting
<b>Defaults</b>	
<b>Sizing</b>	
<b>Quality</b>	
Check Mesh Qua...	Yes, Errors
<input type="checkbox"/> Target Quality	Default (0.050000)
Smoothing	High
Mesh Metric	Characteristic Length
<input type="checkbox"/> Min	0. m
<input type="checkbox"/> Max	0. m
<input type="checkbox"/> Average	0. m
<input type="checkbox"/> Standard Devi...	0. m
<b>Inflation</b>	
<b>Batch Connections</b>	
<b>Advanced</b>	
<b>Statistics</b>	

The screenshot displays the SolidWorks CAD environment. The main window shows a 3D model of a part with a red base and a grey top section. Dimensions are indicated: a vertical height of 40, a horizontal width of 60, and a depth of 60. The software interface includes a top ribbon with various analysis tools such as Design Study, Interference Detection, Measure, Markup, Mass Properties, Section Properties, Sensor Evaluation, Performance Evaluation, Import Diagnostics, Check, Body Compare, Deviation Analysis, Draft Analysis, Symmetry Check, Zebra Stripes, Undercut Analysis, Thickness Analysis, Check Active Document, Curvature, Parting Line Analysis, and Compare Documents. The left-hand side features a Feature Tree with a list of features including Sensors, Anotaciones, Surface Bodies (3), impresora, huevera, suelo, Material <not specified>, Alzado, Planta, Vista lateral, Origen, Surface-Plane1, Plane1, Surface-Plane2, Surface-Loft1, Surface-Knit1, Fillet1, LPattern1, Surface-Knit2, Plane3, Plane2, Plane4, Surface-Plane3, Surface-Plane4, Surface-Extrude1, Surface-Knit3, Thicken2, Surface-Trim1, and Sketch6. The right-hand side shows the SOLIDWORKS Resources panel with sections for Welcome to SOLIDWORKS, SOLIDWORKS Tools (Property Tab Builder, SOLIDWORKS Rx, Performance Benchmark Test, Compare My Score, Copy Settings Wizard, My Products), Online Resources (3DEXPERIENCE Marketplace, Partner Solutions), and Subscription Services. The bottom status bar indicates 'SOLIDWORKS Education Edition - Instructional Use Only', 'Editing Part', and 'MMGS'. The Windows taskbar at the very bottom shows the system tray with the date '12:21' and '07/09/2021'.

- Resumen.

- Diseño de un defo para tener unas fuerzas de unos 15[kN]
- Cálculos con muchos espesores para determinar el óptimo.
- Cálculo con un solo defo para ver el espesor óptimo en cálculo rápido.

## S05t.- Automatización.

Mejora 2122....

- Repaso última sesión

Cálculo de una huevera para poder frenar un paquete de manera plástica obteniendo un espesor óptimo.

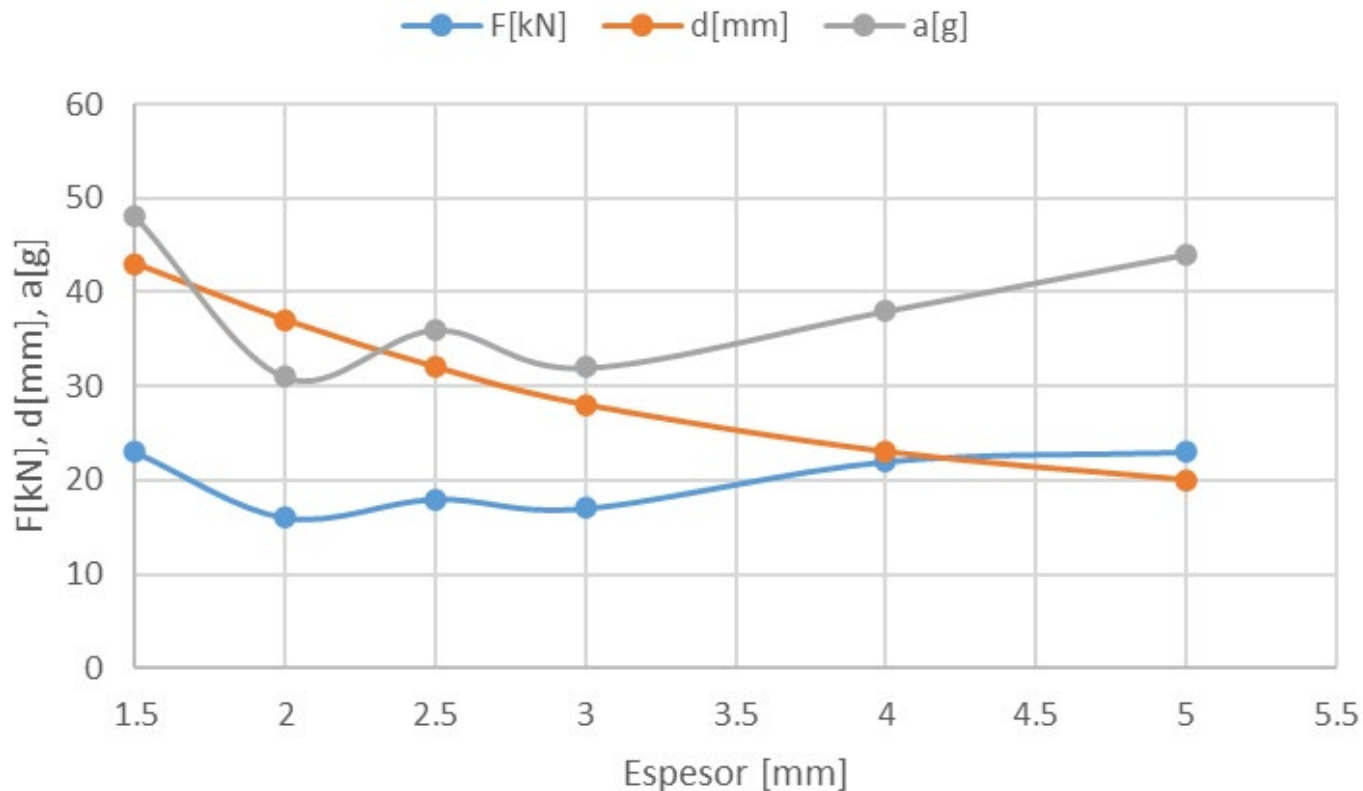
Realización de un cálculo sencillo de una única huevera para poder ir rápido en la optimización

Recordar:

Para el modelo sencillo poner  $3.5[\text{m/s}]$  pero la masa  $50/18=2.778[\text{kg}]$

## • Automatización

Hemos ido realizando simulaciones variando el espesor hasta tener el espesor óptimo. La idea es poder optimizar este proceso para llegar al mejor espesor o diseño de una manera automática.





# • Parámetro input espesor

Outline

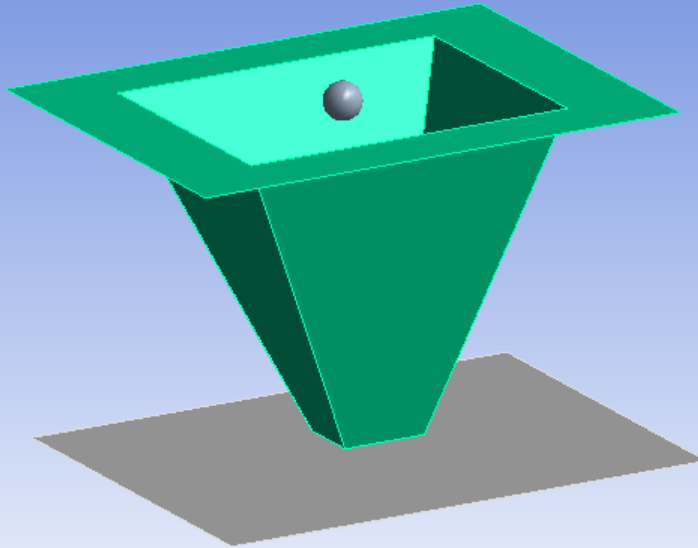
Name Search Outline

Project\*

Model (B4)

- Geometry
  - suelo
  - huevera
- Point Mass
- Materials
- Coordinate Systems
- Connections
- Mesh
- Explicit Dynamics (B5)
  - Initial Conditions
  - Analysis Settings
  - Fixed Support
- Solution (B6)
  - Solution Information
  - Equivalent Plastic Strain
  - Energy Probe
  - Acceleration Probe
  - Deformation Probe
  - Force Reaction

huevera  
09/09/2021 12:43



0.000 0.045 0.090 (m)

0.022 0.068

Details of "huevera"

Graphics Properties

Definition	
Suppressed	No
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
Thickness	2.5e-003 m
Thickness Mode	Manual
Offset Type	Middle
Material	
Assignment	yield2-pulp

Messages

Text	Association	Timestamp
Warning: The specified defeaturing tolerance was found to be larger than a specified size. The ele	Project>Model>Geometry>Huevera-uno-FreeParts Bodies:	Tuesday, September 7, 2021 12:26:22 PM
Warning: The specified defeaturing tolerance was found to be larger than a specified size. The ele	Project>Model>Geometry>Huevera-uno-FreeParts Bodies:	Tuesday, September 7, 2021 12:26:22 PM

# • Parámetro output aceleración

Context B: huevera-sola-y-suelo - Mechanical [ANSYS Mechanical Enterprise]

File Home Result Display Selection Automation

Object Generator Run Macro... Scripting App Store Scripting Help Manage Tools Mechanical Support User Buttons

Outline

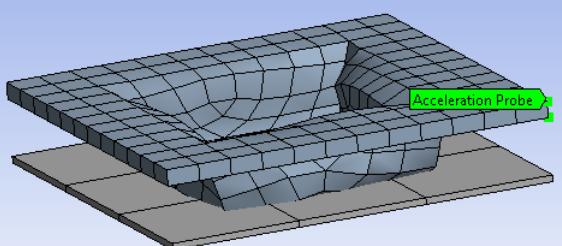
Name Search Outline

Project\*

Model (B4)

- Geometry
  - suelo
  - huevera
  - Point Mass
- Materials
- Coordinate Systems
- Connections
- Mesh
- Explicit Dynamics (B5)
  - Initial Conditions
  - Analysis Settings
  - Fixed Support
- Solution (B6)
  - Solution Information
  - Equivalent Plastic Strain
  - Energy Probe
  - Acceleration Probe
  - Deformation Probe
  - Force Reaction

B: huevera-sola-y-suelo  
Acceleration Probe  
09/09/2021 12:44



Acceleration Probe

0.000 0.045 0.090 (m)

0.022 0.068

Details of "Acceleration Probe"

Definition

Type	Acceleration
Location Method	Geometry Selection
Geometry	1 Vertex
Orientation	Global Coordinate System
Suppressed	No

Options

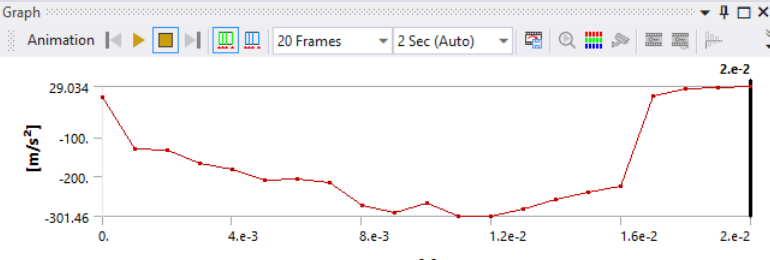
Result Selection	Z Axis
Display Time	End Time
Spatial Resolution	Use Maximum

Results

Maximum Value Over Time	
Z Axis	29.034 m/s <sup>2</sup>
Minimum Value Over Time	
Z Axis	-301.46 m/s <sup>2</sup>

Graph

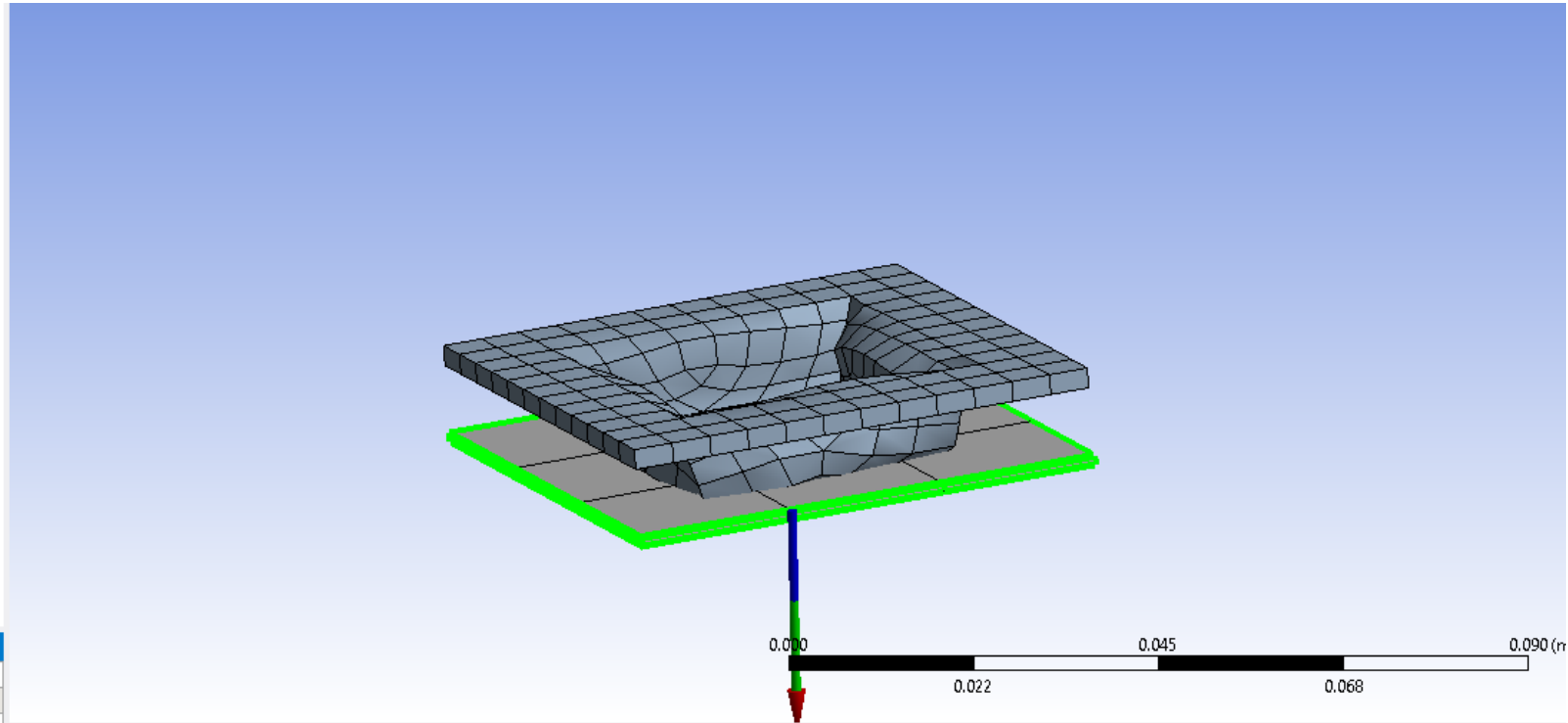
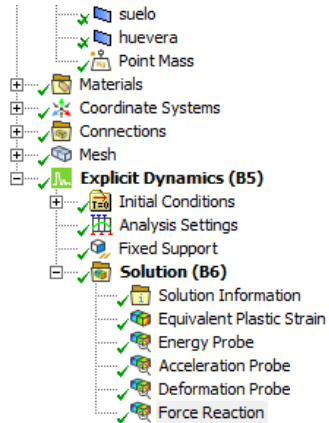
Animation 20 Frames 2 Sec (Auto)



Tabular Data

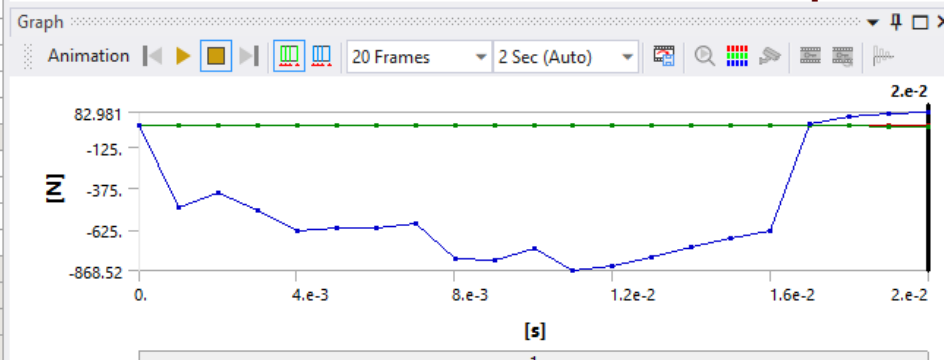
Time [s]	Acceleration Probe (Z) [m/s <sup>2</sup> ]
1	1.1755e-038 0.
2	1.0008e-003 -131.31
3	2.0007e-003 -133.24
4	3.0007e-003 -167.44
5	4.001e-003 -183.13
6	5.0012e-003 -210.48
7	6.0001e-003 -207.31
8	7.0013e-003 -215.33
9	8.0016e-003 -272.78
10	9.0005e-003 -291.16
11	1.0001e-002 -266.55

# • Parámetro output fuerza



Details of "Force Reaction"

Definition	
Type	Force Reaction
Location Method	Boundary Condition
Boundary Condition	Fixed Support
Orientation	Global Coordinate System
Suppressed	No
Options	
Result Selection	All
<input type="checkbox"/> Display Time	End Time
Results	
Maximum Value Over Time	
Minimum Value Over Time	
<input type="checkbox"/> X Axis	-3.6742e-002 N
<input type="checkbox"/> Y Axis	-3.0847 N
<input checked="" type="checkbox"/> Z Axis	-868.52 N
<input type="checkbox"/> Total	0. N



Tabular Data			
Time [s]	<input checked="" type="checkbox"/> Force Reaction (X) [N]	<input checked="" type="checkbox"/> Force Reaction (Y) [N]	<input checked="" type="checkbox"/> Force Reaction (Z) [N]
1	1.1755e-038	0.	0.
2	1.00008e-003	1.2469e-003	-2.8515e-003
3	2.0007e-003	5.2475e-003	-1.7332e-002
4	3.0007e-003	6.4773e-002	4.6746e-002
5	4.001e-003	-3.6742e-002	-9.8522e-003
6	5.0012e-003	5.3319e-002	0.16531
7	6.0001e-003	5.8061e-002	4.6879
8	7.0013e-003	0.31552	1.1107e-002
9	8.0016e-003	0.59346	0.16518
10	9.0005e-003	0.40785	5.2067e-002
11	1.0001e-002	0.54916	-1.7556e-002
12	1.1001e-002	0.33454	9.092e-002

## • Crear imágenes

ANSYS 2021 R1 ACADEMIC

File Home Result Display Selection Automation

Context: B: huevera-sola-y-suelo - Mechanical [ANSYS Mechanical Enterprise]

1.0 (True Scale)

Figure: Perform a screen capture of the Geometry window and place it (as a Figure object) beneath the currently selected object. The orientation, position, zoom, etc. of the Figure can be changed in the Geometry window. Press F1 for help.

Project\* Model (B4)

- Geometry
  - suelo
  - huevera
  - Point Mass
- Materials
- Coordinate Systems
- Connections
- Mesh
- Explicit Dynamics (B5)
  - Initial Conditions
  - Analysis Settings
  - Fixed Support
- Solution (B6)
  - Solution Information
  - Equivalent Plastic Strain
  - Directional Deformation
  - Energy Probe
  - Acceleration Probe
  - Deformation Probe
  - Force Reaction

Equivalent Plastic Strain

Type: Equivalent Plastic Strain - Top/Bottom  
Unit: m/m  
Time: 2.e-002  
Cycle Number: 12606  
09/09/2021 12:48

1.4976 Max

0 Min

Directional Deformation

Type: Directional Deformation(Z Axis)  
Unit: m  
Global Coordinate System  
Time: 2.e-002  
Cycle Number: 12606  
09/09/2021 12:48

0.031647 Max

-1.0359e-5 Min

Graph: Animation | 20 Frames | 2 Sec (Auto)

Tabular Data

Time [s]	Minimum [m/m]	Maximum [m/m]	Average [m/m]
1	1.1755e-038	0.	0.
2	1.0008e-003	0.16015	1.236e-002
3	2.0007e-003	0.61969	4.8583e-002
4	3.0007e-003	1.0154	8.7322e-002
5	4.001e-003	1.4106	0.12898
6	5.0012e-003	1.4676	0.17669
7	6.0001e-003	1.4763	0.21915
8	7.0013e-003	1.4901	0.26685
9	8.0016e-003	1.4969	0.31451
10	9.0005e-003	1.4977	0.36718
11	1.0001e-002	1.4977	0.41286
12	1.1001e-002	1.4976	0.45359
13	1.2e-002	1.499	0.48872
14	1.3001e-002	1.499	0.51716

Details of "Equivalent Plastic Strain"

Scope

Scoping Method: Geometry Selection

Geometry: All Bodies

Position: Top/Bottom

Definition

Type: Equivalent Plastic Strain

By: Time

Display Time: Last

Calculate Time History: Yes

Integration Point Results

Display Option: Averaged

Average Across Bodies: No

Results

Minimum: 0. m/m

Maximum: 1.4976 m/m

Average: 0.55461 m/m

## • Crear imágenes

ANSYS 2021 R1 ACADEMIC

File Home Result Display Selection Automation

Context

B: huevera-sola-y-suelo - Mechanical [ANSYS Mechanical Enterprise]

Quick Launch

Image to File...

Image to File... Save a screen capture of the Geometry window as an image file. This option displays a dialog to make specifications about the file, such as resolution. Supported files include: PNG (.png), JPEG (.jpg), TIFF (.tif), BMP (.bmp), and EPS (.eps).

Press F1 for help.

Project\*

Model (B4)

- Geometry
  - suelo
  - huevera
  - Point Mass
- Materials
- Coordinate Systems
- Connections
- Mesh
- Explicit Dynamics (B5)
  - Initial Conditions
  - Analysis Settings
  - Fixed Support
- Solution (B6)
  - Solution Information
  - Equivalent Plastic Strain
    - Figure
    - Directional Deformation
    - Energy Probe
    - Acceleration Probe
    - Deformation Probe
    - Force Reaction

Equivalent Plastic Strain

B: huevera-sola-y-suelo

Equivalent Plastic Strain

Type: Equivalent Plastic Strain

Unit: m/m

Time: 2.e-002

Cycle Number: 12606

09/09/2021 12:50

1.4976 Max

1.3312

1.1648

0.99839

0.83199

0.6656

0.4992

0.3328

0.1664

0 Min

Directional Deformation

B: huevera-sola-y-suelo

Directional Deformation

Type: Directional Deformation(Z Axis)

Unit: m

Global Coordinate System

Time: 2.e-002

Cycle Number: 12606

09/09/2021 12:48

0.031647 Max

0.02813

0.024612

0.021095

0.017577

0.01406

0.010542

0.0070247

0.0035072

-1.0359e-5 Min

0.000 0.020 0.040 (m)

0.010 0.030

0.000 0.020 0.040 (m)

0.010 0.030

Details of "Equivalent Plastic Strain"

Scope

Scoping Method: Geometry Selection

Geometry: All Bodies

Position: Top/Bottom

Definition

Type: Equivalent Plastic Strain

By: Time

Display Time: Last

Calculate Time History: Yes

Identifier:

Suppressed: No

Integration Point Results

Display Option: Averaged

Average Across Bodies: No

Results

Minimum: 0. m/m

Maximum: 1.4976 m/m

Average: 0.55461 m/m

Graph

Animation

20 Frames

2 Sec (Auto)

1.4992

1.0

0.5

0

0

4.e-3

8.e-3

1.2e-2

1.6e-2

2.e-2

[m/m]

[s]

1

Tabular Data

Time [s]	Minimum [m/m]	Maximum [m/m]	Average [m/m]
1	1.1755e-038	0.	0.
2	1.0008e-003	0.16015	1.236e-002
3	2.0007e-003	0.61969	4.8583e-002
4	3.0007e-003	1.0154	8.7322e-002
5	4.001e-003	1.4106	0.12898
6	5.0012e-003	1.4676	0.17669
7	6.0001e-003	1.4763	0.21915
8	7.0013e-003	1.4901	0.26685
9	8.0016e-003	1.4969	0.31451
10	9.0005e-003	1.4977	0.36718
11	1.0001e-002	1.4977	0.41286
12	1.1001e-002	1.4976	0.45359
13	1.2e-002	1.499	0.48872
14	1.3001e-002	1.499	0.51716

# • Parámetros en Workbench

WB 05-automatizacion - Workbench

File View Tools Units Extensions Jobs Help

Project

Import... Reconnect Refresh Project Update Project Resume Update All Design Points ACT Start Page

Toolbox

Analysis Systems

- Coupled Field Harmonic
- Coupled Field Modal
- Coupled Field Static
- Coupled Field Transient
- Eigenvalue Buckling
- Electric
- Explicit Dynamics
- Fluid Flow - Blow Molding (Polyflo)
- Fluid Flow - Extrusion (Polyflow)
- Fluid Flow (CFX)
- Fluid Flow (Fluent)
- Fluid Flow (Polyflow)
- Harmonic Acoustics
- Harmonic Response
- IC Engine (Fluent)
- Magnetostatic
- Modal
- Modal Acoustics
- Random Vibration
- Response Spectrum
- Rigid Dynamics
- Static Acoustics

Project Schematic

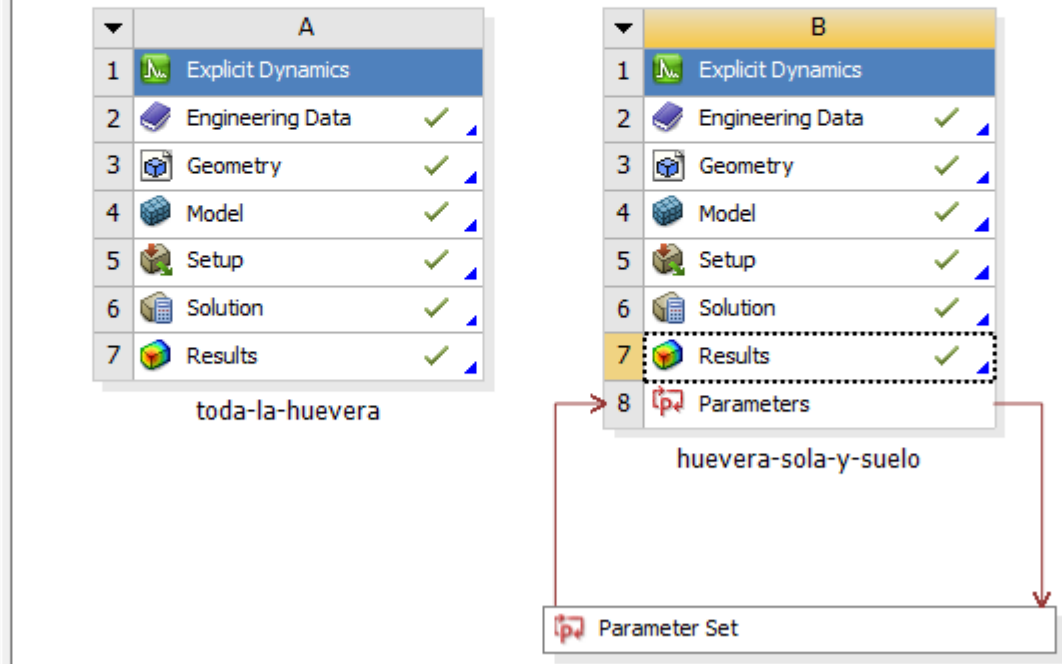
A	
1	Explicit Dynamics
2	Engineering Data ✓
3	Geometry ✓
4	Model ✓
5	Setup ✓
6	Solution ✓
7	Results ✓

toda-la-huevera

B	
1	Explicit Dynamics
2	Engineering Data ✓
3	Geometry ✓
4	Model ✓
5	Setup ✓
6	Solution ✓
7	Results ✓
8	Parameters ✓

huevera-sola-y-suelo

Parameter Set



# • Escenarios de diseño

05-automatizacion - Workbench

File Edit View Tools Units Extensions Jobs Help



Project Parameter Set x

Resume Update All Design Points

Toolbox

Update All Design Points

No toolbox items are applicable for the current selection.

	A	B	C	D
1	ID	Parameter Name	Value	Unit
2	Input Parameters			
3	huevera-sola-y-suelo (B1)			
4	P1	huevera Thickness	0.0025	m
*	New input parameter	New name	New expression	
6	Output Parameters			
7	huevera-sola-y-suelo (B1)			
8	P2	Acceleration Probe Minimum Z Axis	-301.46	m s <sup>-2</sup>
9	P3	Force Reaction Minimum Z Axis	-868.52	N
*	New output parameter		New expression	
11	Charts			

Properties of Design Points: Parameter Set

	A	B
1	Property	Value
2	Design Point Report	
3	Report Image	None

Table of Design Points

	A	B	C	D	E	F	G
1	Name	P1 - huevera Thickness	P2 - Acceleration Probe Minimum Z Axis	P3 - Force Reaction Minimum Z Axis	Ret...	Retained Data	Note
2	Units	m	m s <sup>-2</sup>	N			
3	DP 0 (Current)	0.0025	-301.46	-868.52	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
4	DP 1	0.001	⚡	⚡	<input type="checkbox"/>		
5	DP 2	0.002	⚡	⚡	<input type="checkbox"/>		
6	DP 3	0.003	⚡	⚡	<input type="checkbox"/>		
7	DP 4	0.004	⚡	⚡	<input type="checkbox"/>		
8	DP 5	0.005	⚡	⚡	<input type="checkbox"/>		
*					<input type="checkbox"/>		

Chart: No data

# • Escenarios de diseño

05-automatizacion - Workbench

File View Tools Units Extensions Jobs Help

Project Parameter Set

Resume Update All Design Points

Toolbox

No toolbox items are applicable for the current selection.

Outline of All Parameters

	A	B	C	D
1	ID	Parameter Name	Value	Unit
2	Input Parameters			
3	huevera-sola-y-suelo (B1)			
4	P1	huevera Thickness	0.002	m
*	New input parameter	New name	New expression	
6	Output Parameters			
7	huevera-sola-y-suelo (B1)			
8	P2	Acceleration Probe Minimum Z Axis	-215.31	m s <sup>-2</sup>
9	P3	Force Reaction Minimum Z Axis	-614.63	N
*	New output parameter	New expression		
11	Charts			

Properties of Design Points: Parameter Set

	A	B
1	Property	Value
2	Design Point Report	
3	Report Image	None

Table of Design Points

	A	B	C	D	E	F	G
1	Name	P1 - huevera Thickness	P2 - Acceleration Probe Minimum Z Axis	P3 - Force Reaction Minimum Z Axis	Ret...	Retained Data	Note
2	Units	m	m s <sup>-2</sup>	N			
3	DP 0	0.0025	-301.46	-868.52	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
4	DP 1	0.001	-83.6	-11548	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
5	DP 2 (Current)	0.002	-215.31	-614.63	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
6	DP 3	0.003	-360.3	-999.45	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
7	DP 4	0.004	-416.09	-1150.7	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
8	DP 5	0.005	-465.65	-1305.3	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
*							

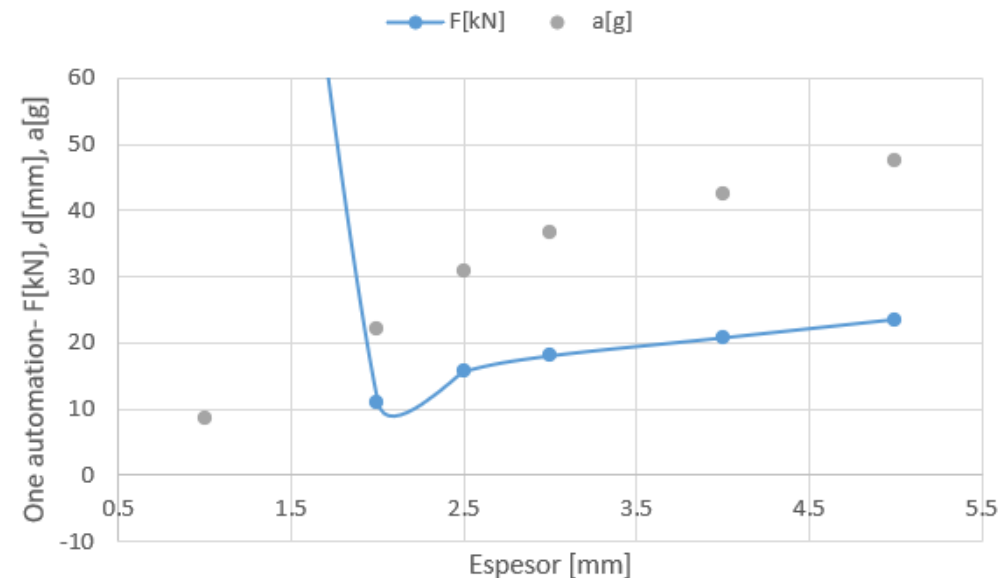
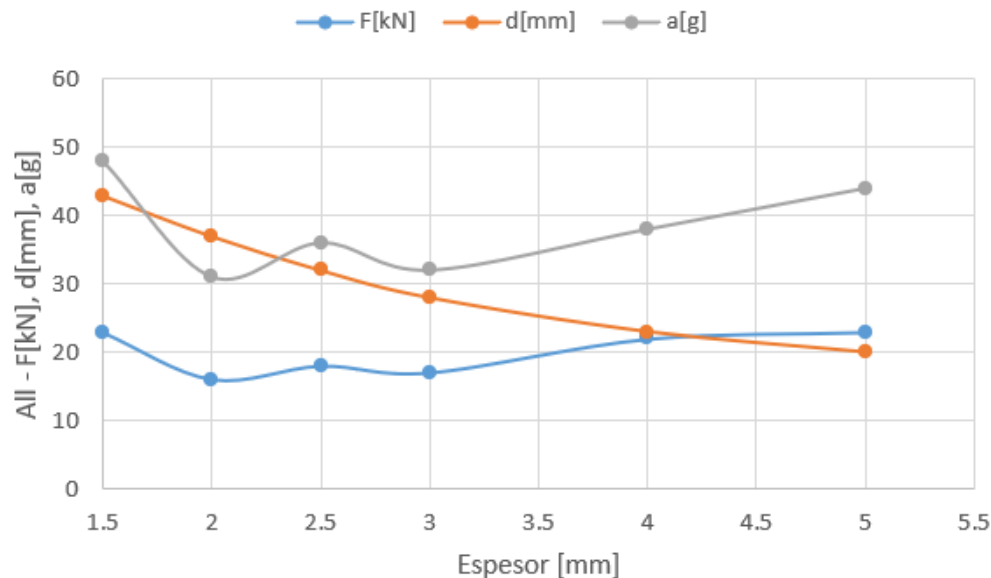
Chart: No data

- Set Update Order by Row
- Show Update Order
- Optimize Update Order
- Set as Current**
- Copy inputs to Current
- Delete Design Point
- Duplicate Design Point
- Export Selected Design Points
- Update Selected Design Points
- Export Table Data as CSV
- Import from Design Point Service



## • Resultados comparados

Modelo de 18 sin automatizar con modelo de sólo 1 automatizado (La fuerza del gráfico es la F de cálculo multiplicada x18 para comparar). Ahora la aceleración baja para espesores pequeños pues no llega a acabar el cálculo y ver el segundo pico. Cuidado con mínimo y máximo!!



# • Comprobación de resultados

Es necesario mirar los resultados por si hay algún problema. Con 2[mm] hay un poco de Hourglass pero OK.

Model (B4)

- Geometry
  - suelo
  - huevera
  - Point Mass
- Materials
- Coordinate Systems
- Connections
- Mesh
- Explicit Dynamics (B5)
  - Initial Conditions
  - Analysis Settings
  - Fixed Support
- Solution (B6)
  - Solution Information
  - Equivalent Plastic Strain
  - Figure
  - Directional Deformation
  - Energy Probe
  - Acceleration Probe
  - Deformation Probe
  - Force Reaction

Details of "Energy Probe"

Definition

Type	Energy
Geometry	2 Bodies
Suppressed	No

Options

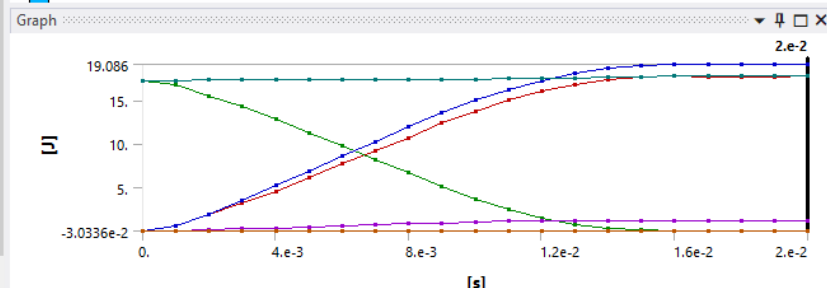
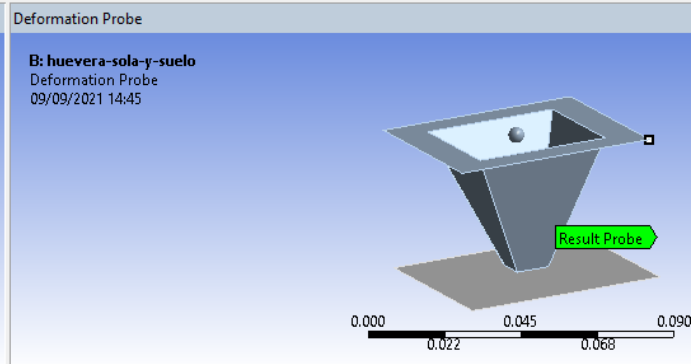
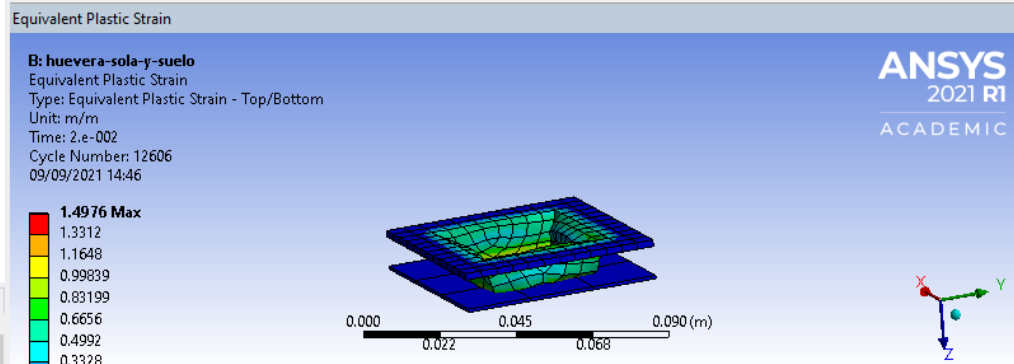
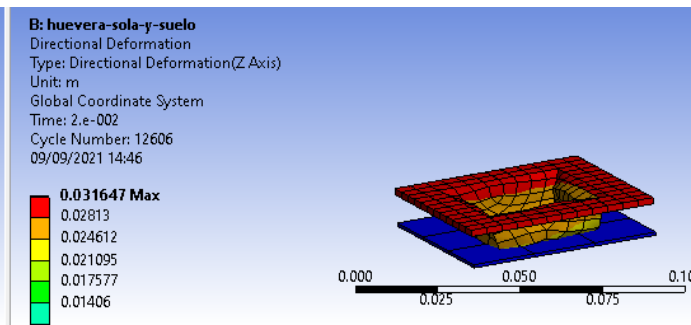
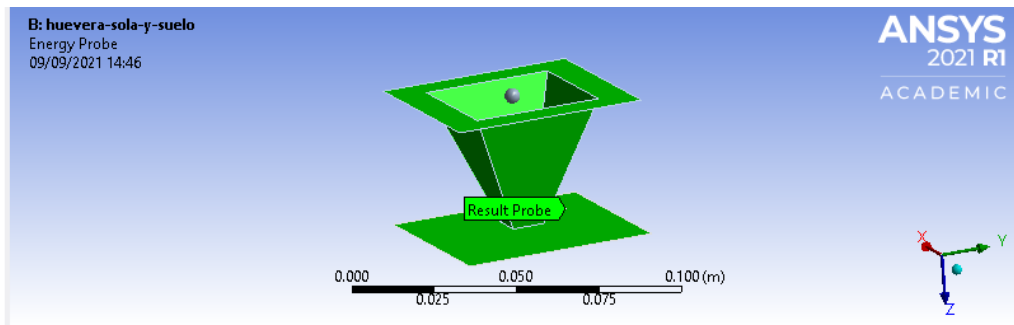
Result Selection: All

Display Time: End Time

Results

Maximum Value Over Time

- Internal
- Kinetic
- Plastic Work
- Hourglass
- Contact
- Total



Tabular Data

Time [s]	Energy Probe (Internal) [J]	Energy Probe (Kinetic) [J]	Energy Probe (Plastic Work) [J]	Energy Probe (Hourglass) [J]
1	1.1755e-038	0.	17.223	0.
2	1.0008e-003	0.4917	16.725	0.5052
3	2.0007e-003	1.8343	15.436	1.9164
4	3.0007e-003	3.1412	14.199	3.4728
5	4.001e-003	4.4902	12.864	5.1666
6	5.0012e-003	6.1605	11.168	6.8762
7	6.0001e-003	7.6313	9.7079	8.5191
8	7.0013e-003	9.2286	8.0989	10.203
9	8.0016e-003	10.671	6.6702	11.865
10	9.0005e-003	12.309	5.0295	13.487
11	1.0001e-002	13.722	3.6456	14.935

# • Comprobación de resultados

Con 1[mm] se disparan las energías y el cálculo no es válido.

**Project\***

- Model (B4)
  - Geometry
    - suelo
    - huevera
    - Point Mass
  - Materials
  - Coordinate Systems
  - Connections
  - Mesh
  - Explicit Dynamics (B5)
    - Initial Conditions
    - Analysis Settings
    - Fixed Support
  - Solution (B6)
    - Solution Information
    - Equivalent Plastic Strain
    - Directional Deformation
    - Energy Probe**
    - Acceleration Probe
    - Deformation Probe
    - Force Reaction

---

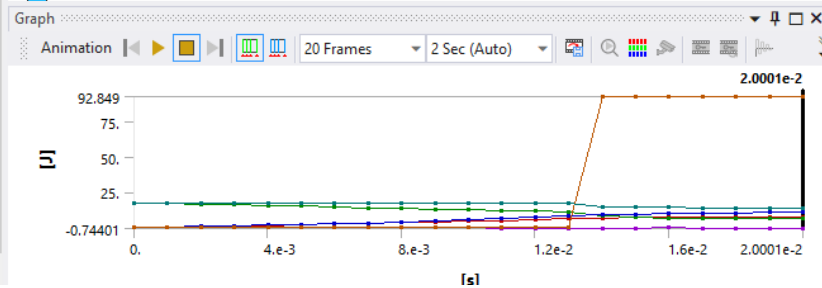
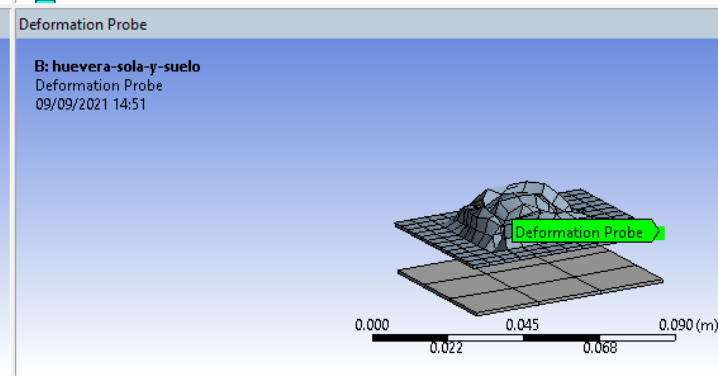
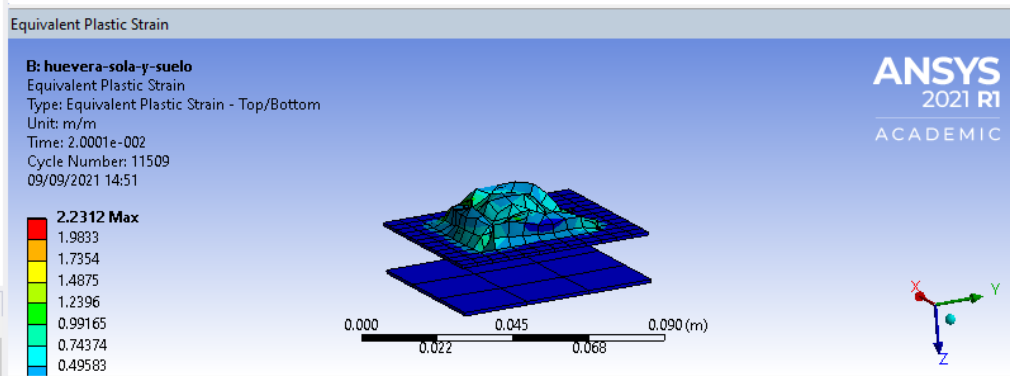
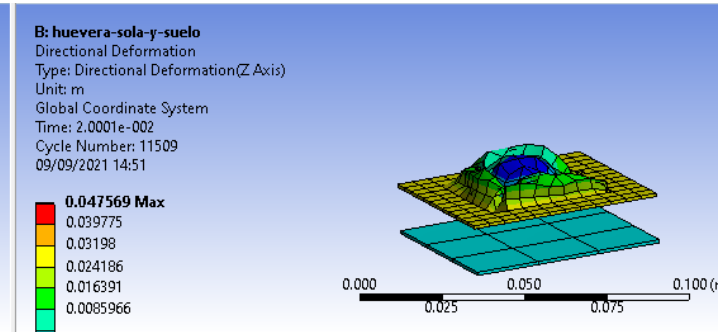
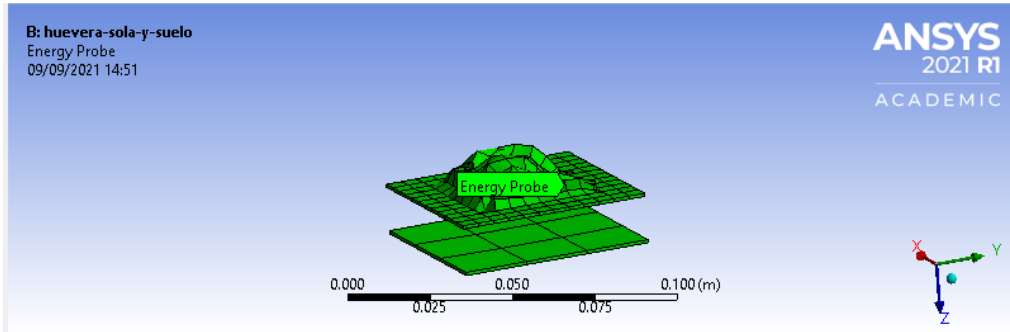
Properties of "Energy Probe"

Property	Value
Type	Energy
Geometry	2 Bodies
Compressed	No

---

Results

Category	Value
Internal	7.5659 J
Kinetic	17.106 J
Plastic Work	10.832 J
Hourglass	0.184 J
Contact	92.849 J
Total	17.132 J

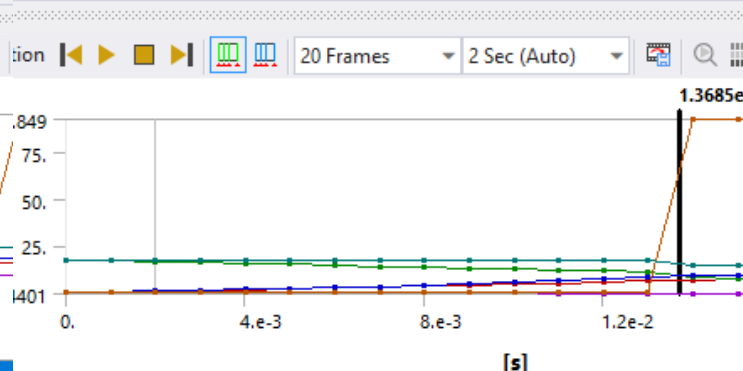
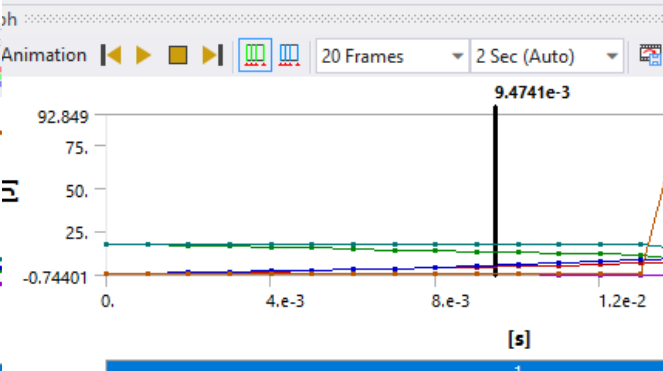
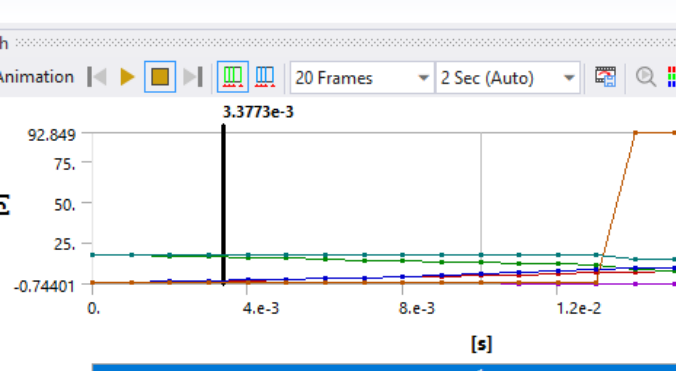
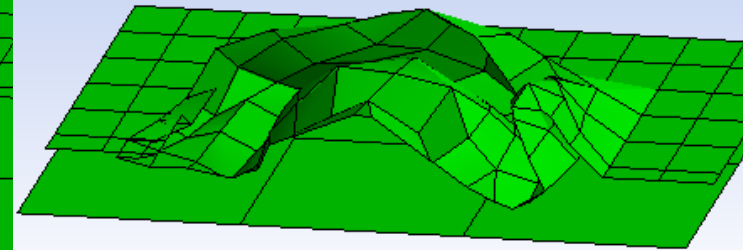
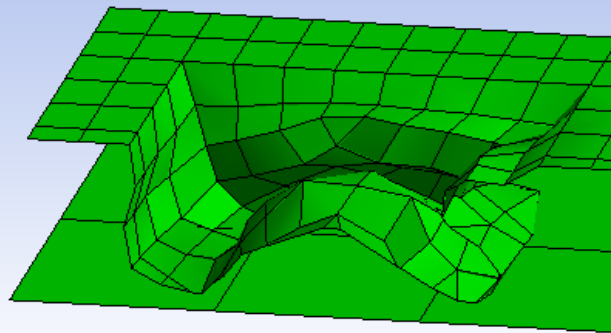
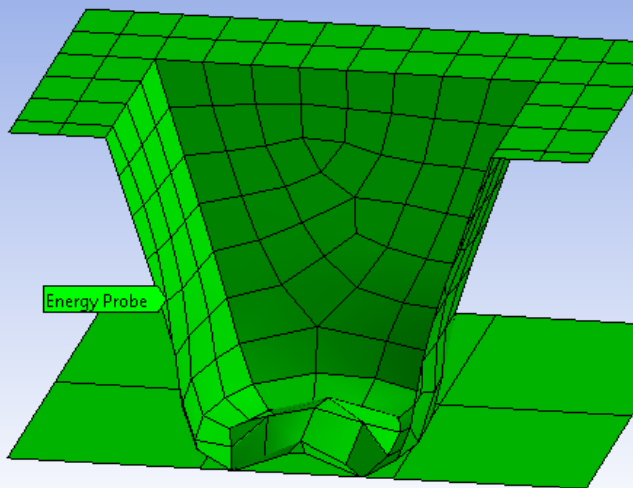


Tabular Data

Time [s]	Energy Probe (Internal) [J]	Energy Probe (Kinetic) [J]	Energy Probe (Plastic Work) [J]	Energy Probe (Hourglass) [J]
1	1.1755e-038	0.	17.106	0.
2	1.0005e-003	0.19732	16.906	0.2043
3	2.0001e-003	0.68711	16.427	0.69857
4	3.0006e-003	1.0319	16.087	1.0595
5	4.0009e-003	1.443	15.675	1.5103
6	5.0004e-003	1.8749	15.241	1.9888
7	6.0016e-003	2.4538	14.664	2.6101
8	7.0015e-003	3.0323	14.086	3.211
9	8.0001e-003	3.6229	13.509	3.9219
10	9.0003e-003	4.0441	13.029	4.8127
11	1.0001e-002	4.4875	12.601	5.5027

# • Comprobación de resultados

Con 1[mm] se disparan las energías y el cálculo no es válido.



# • Report

Context B: huevera-sola-y-suelo - Mechanical [ANSYS Mechanical Enterprise]

File Home Project Display Selection Automation

My Computer My Computer  
 Distributed  
 Cores 2 Solve

Analysis  
 Coordinate System Comment Section Plane  
 Remote Point Chart Annotation

Units Worksheet Keyframe Animation Wizard Show Errors Unit Converter Tools

Manage Views Selection Information Report Preview Key Assignments  
 Print Preview Report Preview  
 Manage Manage User Defined Reset Layout Layout

Outline Name Search Outline  
 Generate Send To Print Publish Font Size

**Report Preview**  
 Generate a report of the simulation in the Report Preview tab. The report includes extensive information about the analysis, including the Geometry, Materials, Coordinate Systems, Contact, Loads, Solution, etc.  
 Press F1 for help.

**Project\***  
 Model (B4)  
 Geometry  
 suelo  
 huevera  
 Point Mass  
 Materials  
 Coordinate Systems  
 Connections  
 Mesh  
 Explicit Dynamics (B5)  
 Initial Conditions  
 Analysis Settings  
 Fixed Support  
 Solution (B6)  
 Solution Information  
 Equivalent Plastic Strain  
 pe  
 Directional Deformation  
 Energy Probe  
 Acceleration Probe  
 Deformation Probe  
 Force Reaction

1.0308+003
1.413e+009
1.069e+009
4.41e+008
2.62e+008
2.14e+008
1.38e+008
1.14e+008
8.62e+007

10000	0
20000	0
1.e+005	0
2.e+005	0
1.e+006	0

**TABLE 46**  
**yield2-pulp > Strain-Life Parameters**

Strength Coefficient Pa	Strength Exponent	Ductility Coefficient	Ductility Exponent	Cyclic Strength Coefficient Pa	Cyclic Strain Hardening Exponent
9.2e+008	-0.106	0.213	-0.47	1.e+009	0.2

**TABLE 47**  
**yield2-pulp > Isotropic Elasticity**

Young's Modulus Pa	Poisson's Ratio	Bulk Modulus Pa	Shear Modulus Pa	Temperature C
2.e+009	0.3	1.6667e+009	7.6923e+008	

**TABLE 48**  
**yield2-pulp > Isotropic Relative Permeability**

Relative Permeability
10000

**TABLE 49**  
**yield2-pulp > Bilinear Isotropic Hardening**

Yield Strength Pa	Tangent Modulus Pa	Temperature C
2.e+006	0	

Details of "Solution Information"  
**Title Page**  
 Author  
 Subject  
 Prepared for

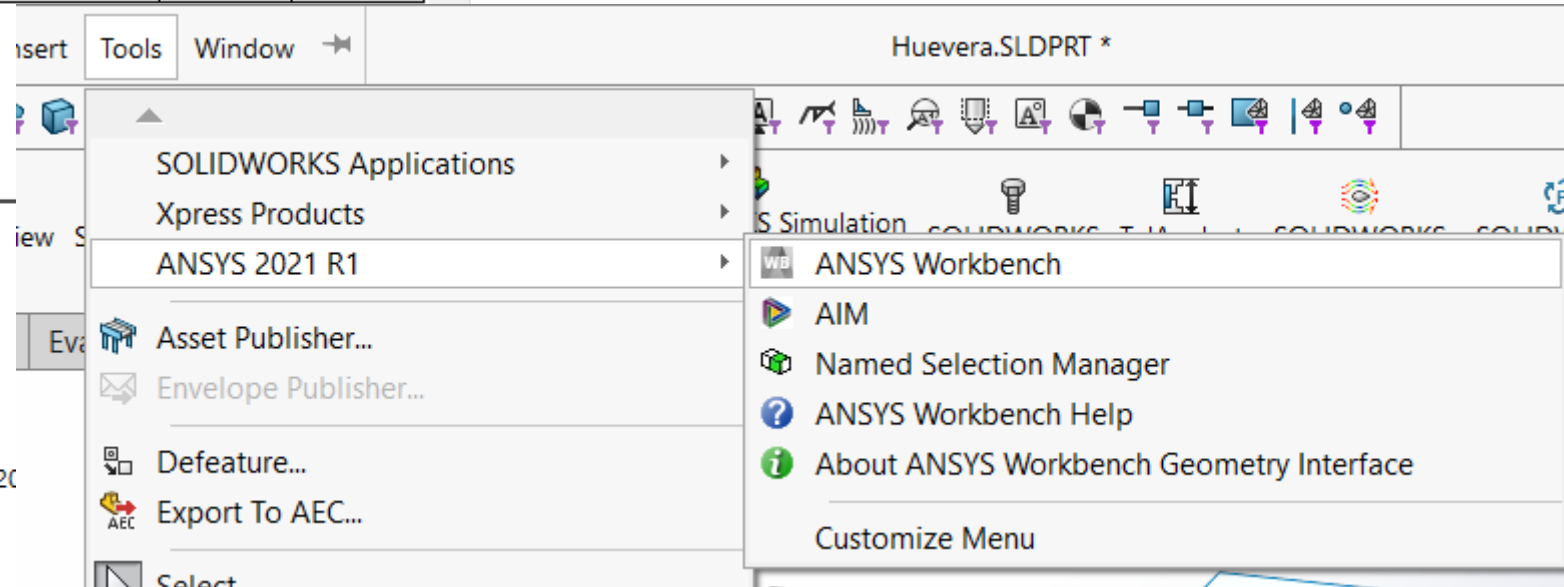
Geometry Report Preview  
 Messages

# • Parámetros geométricos desde SolidWorks

Add-Ins



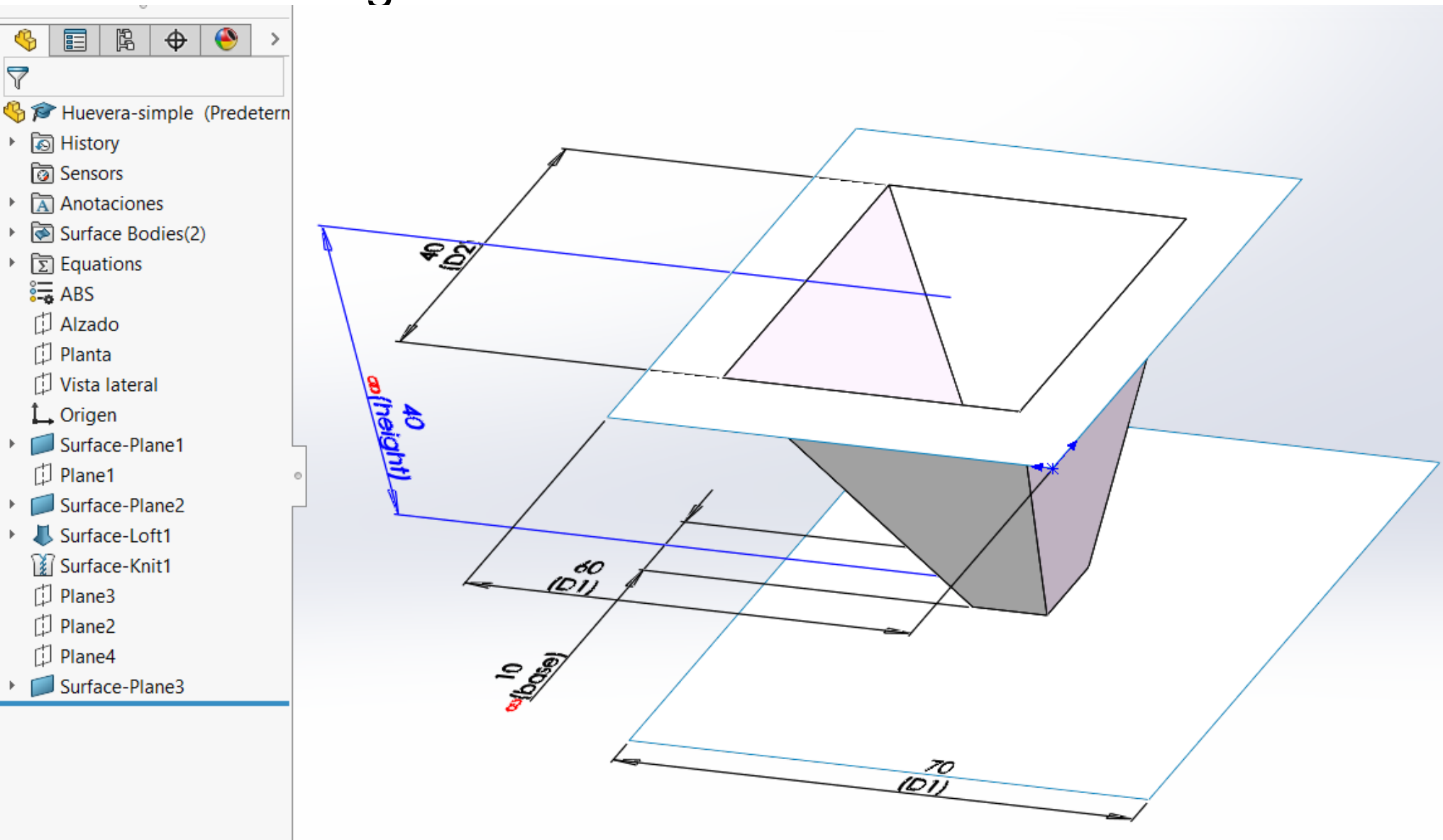
Active Add-ins	Start Up	Last Load Time
<input type="checkbox"/> SOLIDWORKS Utilities		
<input type="checkbox"/> TolAnalyst		
<b>SOLIDWORKS Add-ins</b>		
<input type="checkbox"/> 3DEXPERIENCE Marketplace		
<input type="checkbox"/> Autotrace		
<input type="checkbox"/> SOLIDWORKS CAM 2020		
<input type="checkbox"/> SOLIDWORKS Composer		
<input type="checkbox"/> SOLIDWORKS Electrical		
<input type="checkbox"/> SOLIDWORKS Flow Simulation 20		
<input type="checkbox"/> SOLIDWORKS Forum 2020		
<input type="checkbox"/> SOLIDWORKS Plastics		
<input type="checkbox"/> SOLIDWORKS Visualize		
<b>Other Add-ins</b>		
<input type="checkbox"/> 3DCloudByMe Plug-in	<input type="checkbox"/>	--
<input checked="" type="checkbox"/> Ansys 2021 R1	<input checked="" type="checkbox"/>	< 1s
<input type="checkbox"/> KISSsoftSWAddin	<input type="checkbox"/>	--
<input type="checkbox"/> SOLIDWORKS XPS Driver 2020	<input type="checkbox"/>	--



The screenshot shows the SolidWorks interface with the 'Tools' menu open. The 'ANSYS 2021 R1' sub-menu is expanded, showing the following options:

- ANSYS Workbench
- AIM
- Named Selection Manager
- ANSYS Workbench Help
- About ANSYS Workbench Geometry Interface
- Customize Menu

# • Parámetros geométricos desde SolidWorks



# • Propiedades de material como parámetro

Properties of Outline Row 3: pulp-y2

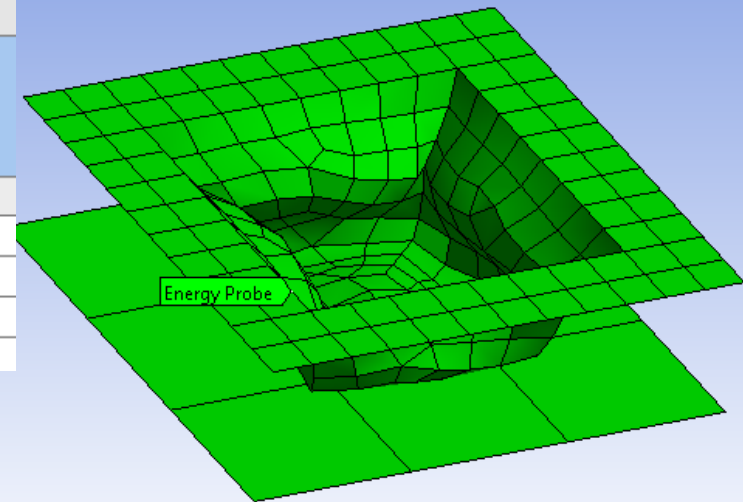
	A	B	C	D	E
1	Property	Value	Unit	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
2	<input checked="" type="checkbox"/> Material Field Variables	<input type="checkbox"/> Table		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
3	<input checked="" type="checkbox"/> Density	2000	kg m <sup>-3</sup>	<input type="checkbox"/>	<input type="checkbox"/>
4	<input type="checkbox"/> <input checked="" type="checkbox"/> Isotropic Elasticity			<input type="checkbox"/>	<input type="checkbox"/>
5	Derive from	Young's Modulus and Poisson...			
6	Young's Modulus	2E+09	Pa	<input type="checkbox"/>	<input type="checkbox"/>
7	Poisson's Ratio	0.3			<input type="checkbox"/>
8	Bulk Modulus	1.6667E+09	Pa		<input type="checkbox"/>
9	Shear Modulus	7.6923E+08	Pa		<input type="checkbox"/>
10	<input type="checkbox"/> <input checked="" type="checkbox"/> Bilinear Isotropic Hardening				
11	Yield Strength	2E+06	Pa		<input checked="" type="checkbox"/>
12	Tangent Modulus	0	Pa	<input type="checkbox"/>	<input type="checkbox"/>
13	<input checked="" type="checkbox"/> Specific Heat Constant Pressure, C <sub>p</sub>	434	J kg <sup>-1</sup> C <sup>-1</sup>	<input type="checkbox"/>	<input type="checkbox"/>



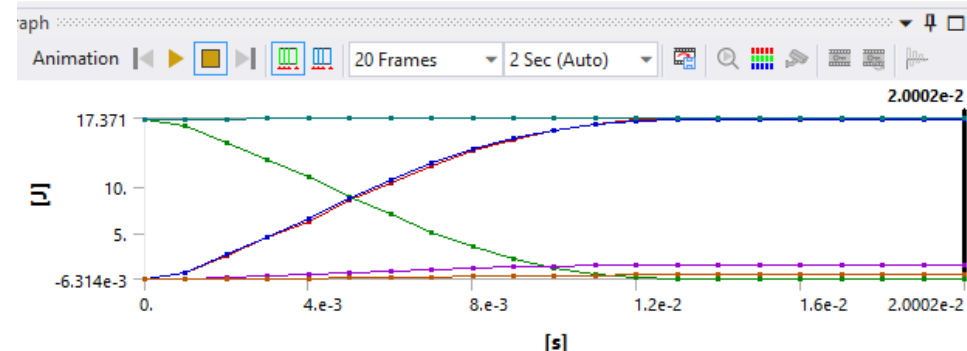
# • Automatización con propiedades de material

Generamos variantes de propiedades de material.

Table of Design Points							
	A	B	C	D	E	F	G
1	Name	P4 - Yield Strength	P5 - Accele... Probe Minimum Z Axis	P6 - Deform... Probe Maximum Z Axis	P7 - Force Reaction Minimum Z Axis	<input type="checkbox"/> Ret...	Retained Data
2	Units	Pa	m s <sup>-2</sup>	m	N		
3	DP 1 (Current)	2E+06	-4306.3	0.042002	-12129	<input checked="" type="checkbox"/>	✓
4	DP 2	4E+06	-335.06	0.025717	-937.38	<input checked="" type="checkbox"/>	✓
5	DP 3	6E+06	-471.13	0.019781	-1318.5	<input checked="" type="checkbox"/>	✓
*						<input type="checkbox"/>	

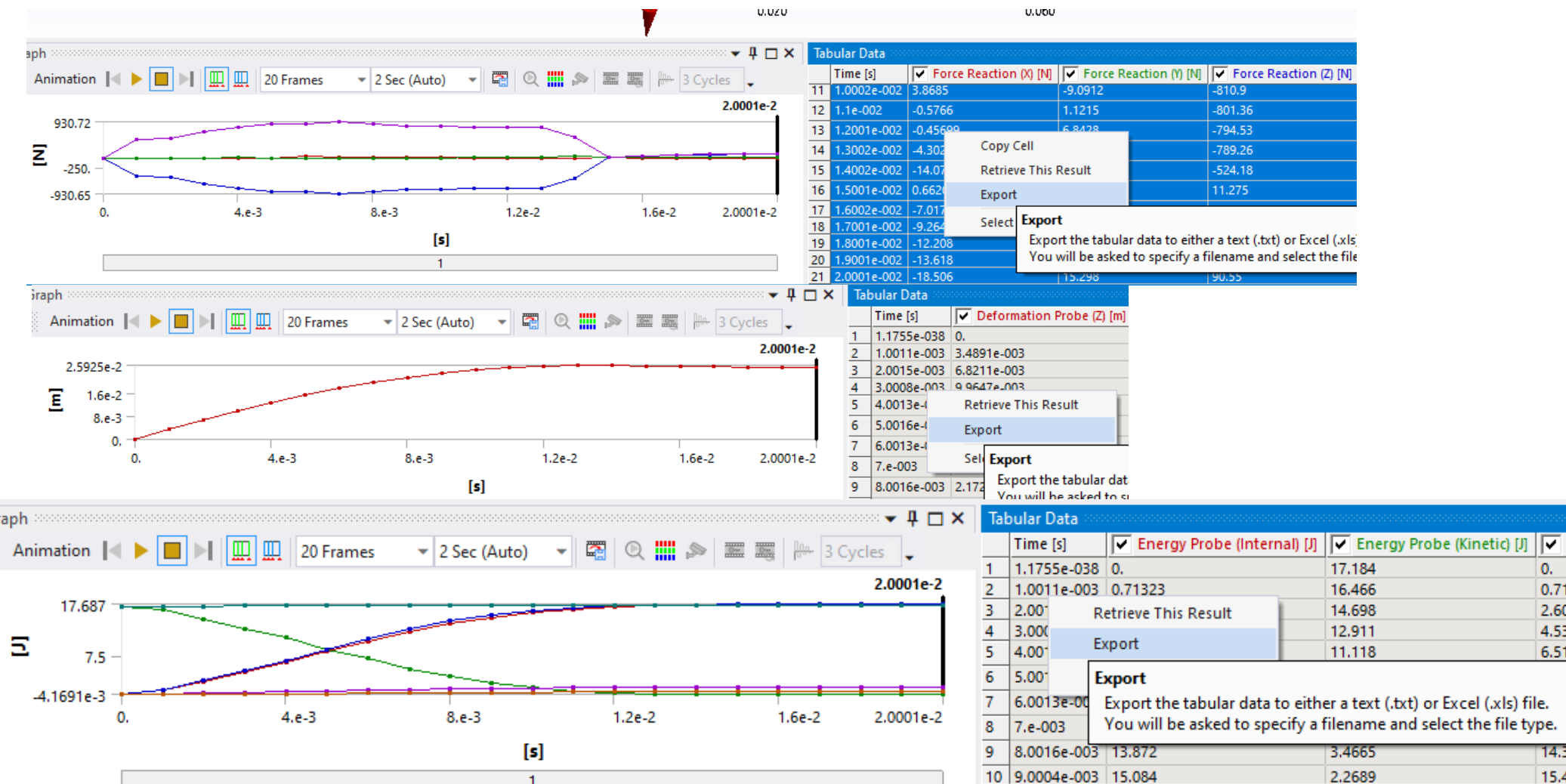


0.000



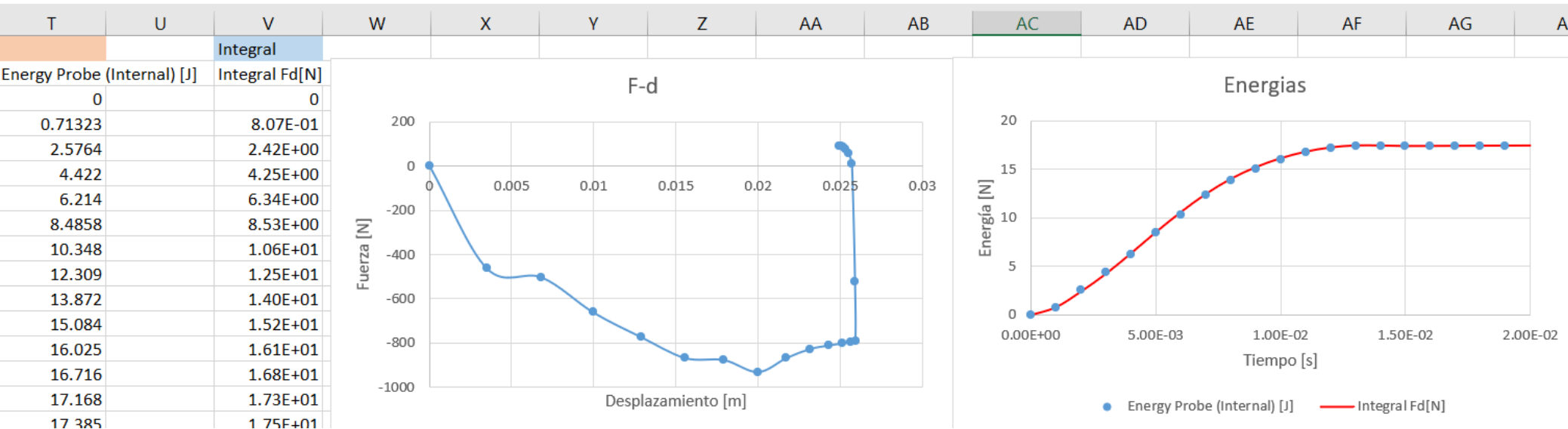
# • Fuerza desplazamiento y energía

Generamos la curva de fuerza y desplazamiento .

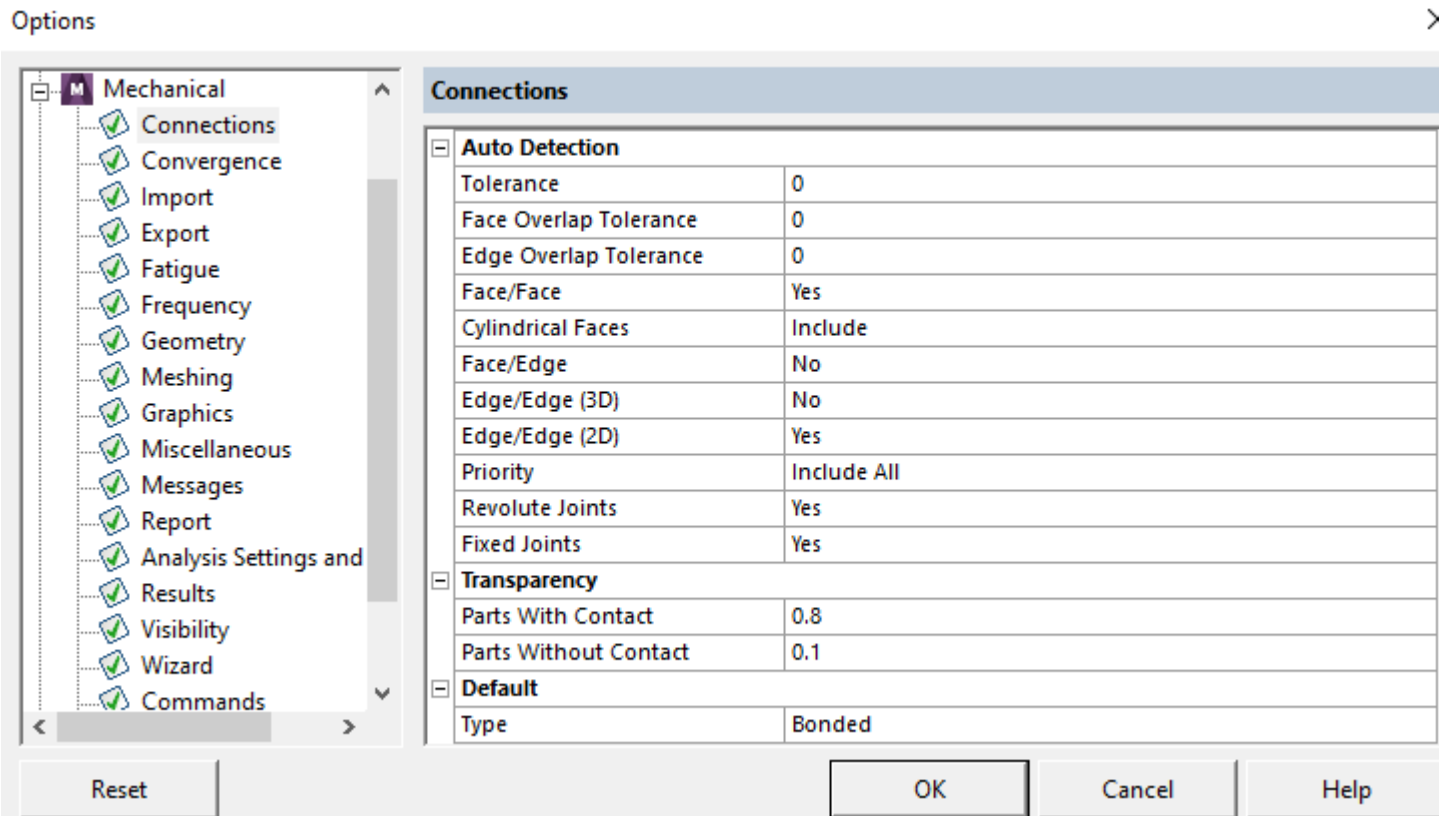


# • Trabajo en Excel

Trabajamos para comparar y practicar a integrar en Excel y ver que la energía es la integral de fuerza-desplazamiento.



# • Mechanical, file, options



# • Scripts

Un buen lugar para mirar scripts es

[https://mapldocs.pyansys.com/user\\_guide/index.html](https://mapldocs.pyansys.com/user_guide/index.html)



Getting Started **User Guide** API Reference MAPDL Commands Examples Contributing



🔍 Search the docs ...

Initial Setup and Launching MAPDL Locally

PyMAPDL Language and Usage

ANSYS APDL Interactive Control Examples

Interactive Plotting

Mesh and Geometry

Post-Processing

Setting and Retrieving Parameters

Translating Scripts

APDL Math Overview

Create a Pool of MAPDL Instances

File Explorer Overview

## User Guide

This guide provides a general overview of the basics and usage of the PyMAPDL library.

## PyMAPDL Basic Overview

The `launch_mapdl()` function within the `ansys-mapdl-core` library creates an instance of `Mapdl` in the background and sends commands to that service. Errors and warnings are processed Pythonically letting the user develop a script real-time without worrying about if it will function correctly when deployed in batch mode.

MAPDL can be started from python in gRPC mode using `launch_mapdl()`. This starts MAPDL in a temporary directory by default. You can change this to your current directory with:

```
import os
from ansys.mapdl.core import launch_mapdl

path = os.getcwd()
mapdl = launch_mapdl(run_location=path)
```

☰ On this page

PyMAPDL Basic Overview

Calling MAPDL Pythonically

# • Bases de datos de materiales

<https://paperonweb.com/pulppro.htm>

GE :Untitled - GRANTA EduPack 2020 - [MaterialUniverse\Hybrids: composites, foams, natural materials\Natural materials]

File Edit View Select Tools Window Feature Request Help




Home Browse Search Chart/Select Solver Eco Audit Synthesizer Learn Tools Settings Help

Search

Database: Level 1 Change...

paper AND pulp

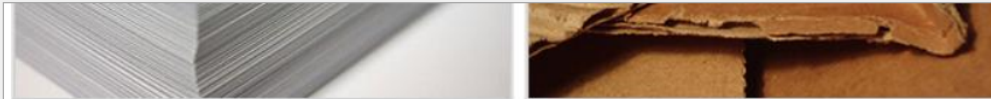
## MaterialUniverse (3)

-  Paper and cardboard
-  Bamboo
-  Titanium alloys

Home Paper and cardboard

## Paper and cardboard

Datasheet view: All properties Show/Hide Find Similar



### Caption

1. Stack of copy paper © Jonathan Joseph Bondhus at en.wikipedia (CC BY-SA 3.0) 2. Corrugated cardboard © Richard Wheeler (Zephyris) at en.wikipedia (CC BY-SA 3.0)

### The material

Papyrus, the forerunner of paper, was made from the flower stem of the reed, native to Egypt; it has been known and used for over 5000 years. Paper, by contrast, is a Chinese invention (105 AD). It is made from pulped cellulose fibers derived from wood, cotton or flax. There are many different types of paper and paper board: tissue paper - newsprint, kraft paper for packaging, office paper, fine glazed writing paper, cardboard - and a correspondingly wide range of properties. The data below span the range of newsprint and kraft paper.

### Composition (summary) ⓘ

Cellulose fibers, usually with filler and colorant

### General properties

Density	ⓘ	700	-	1.15e3	kg/m <sup>3</sup>
Price	ⓘ	* 0.886	-	1.08	EUR/kg

### Mechanical properties

Young's modulus	ⓘ	2	-	4	GPa
Yield strength (elastic limit)	ⓘ	15	-	34	MPa
Tensile strength	ⓘ	15	-	34	MPa
Elongation	ⓘ	* 0.38	-	1.7	% strain
Hardness - Vickers	ⓘ	* 6	-	9	HV
Fatigue strength at 10 <sup>7</sup> cycles	ⓘ	* 13.8	-	24.8	MPa
Fracture toughness	ⓘ	2	-	3	MPa.m <sup>0.5</sup>

### Thermal properties

Maximum service temperature	ⓘ	76.9	-	130	°C
Thermal conductor or insulator?	ⓘ	Good insulator			

# • Bases de datos de materiales

GE :Untitled - GRANTA EduPack 2020 - [MaterialUniverse\Hybrids: composites, foams, honeycombs, natural materials\Composites\Natural material composites\Paper an]

File Edit View Select Tools Window Feature Request Help

Home Browse Search Chart/Select Solver Eco Audit Synthesizer Learn Tools Settings Help

Search Database: Level 3 Change...

paper AND pulp

MaterialUniverse (50)

- Paper and cardboard
- Paperboard**
- Stainless steel, duplex, Ilium PD, cast, water quenched
- Stainless steel, duplex, Ilium PD, cast, water quenched & aged
- Satinwood (l)
- Parana-pine (l)
- Satinwood (t)
- Parana-pine (t)
- Bamboo (longitudinal)
- Bamboo (transverse)
- Bamboo
- Stainless steel, duplex, UNS S33207, annealed
- Palm
- Stainless steel, austenitic, ASTM CH-10, cast, water quenched
- Stainless steel, austenitic, ASTM CN-7MS, cast, water quenched
- Duplex (semi-austenitic) stainless steel
- Stainless steel, duplex, ASTM CD-4MCu, cast, water quenched
- Martensitic stainless steel
- Stainless steel, martensitic, ASTM CA-40, cast, tempered at 300°C
- Stainless steel, martensitic, ASTM CA-40, cast, tempered at 500°C
- Stainless steel, martensitic, ASTM CA-40, cast, tempered at 600°C
- Stainless steel, martensitic, ASTM CA-40, cast, tempered at 700°C
- Stainless steel, martensitic, ASTM CB-7Cu, cast, aged at 480°C
- Stainless steel, martensitic, ASTM CB-7Cu, cast, aged at 495°C
- Stainless steel, martensitic, ASTM CB-7Cu, cast, aged at 550°C
- Stainless steel, martensitic, ASTM CB-7Cu, cast, aged at 580°C
- Stainless steel, martensitic, ASTM CB-7Cu, cast, aged at 595°C
- Stainless steel, martensitic, ASTM CB-7Cu, cast, aged at 620°C

Paperboard

Datasheet view: All attributes Show/Hide Find Similar

Cellulose

Form	Other
Material family	Natural
Base material	Cellulose
Renewable content	85 %

Price

Price	* 0.671	- 1.34	EUR/kg
Price per unit volume	* 638	- 1.34e3	EUR/m <sup>3</sup>

Physical properties

Density	950	- 1e3	kg/m <sup>3</sup>
---------	-----	-------	-------------------

Mechanical properties

Yield strength (elastic limit)	20	- 50	MPa
Tensile strength	20	- 50	MPa
Specific strength	20.5	- 51.3	kN.m/kg
Elongation	* 2.2	- 4	% strain
Compressive strength	* 20	- 50	MPa
Bulk modulus	* 0.1	- 0.5	GPa
Poisson's ratio	* 0.3	- 0.4	
Fatigue strength at 10 <sup>7</sup> cycles	* 10	- 20	MPa

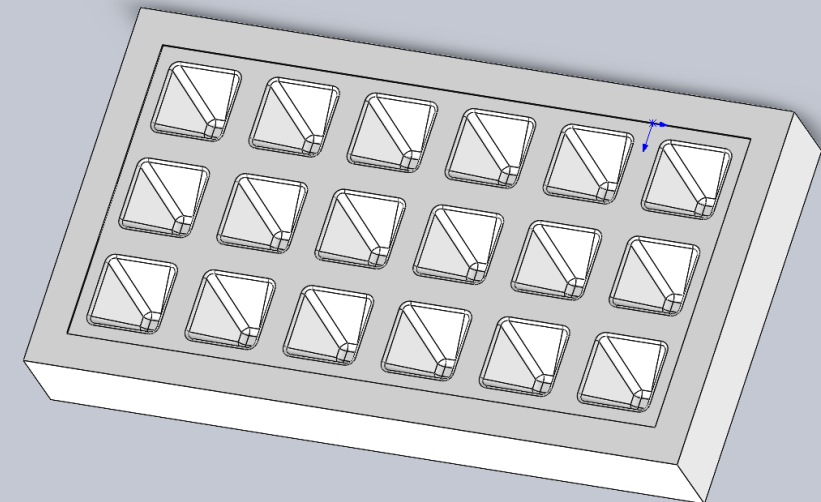
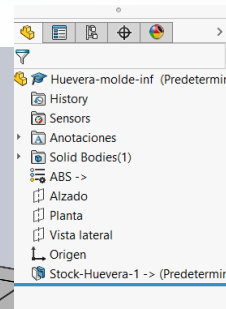
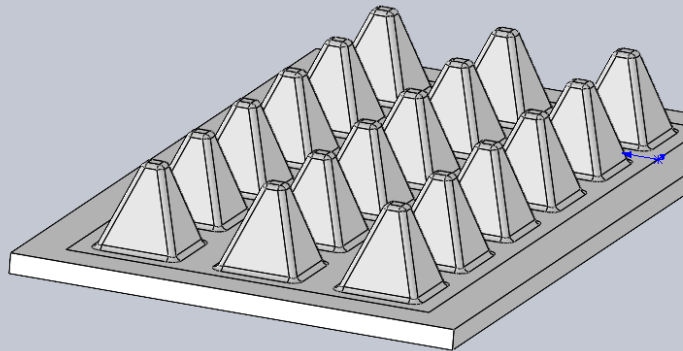
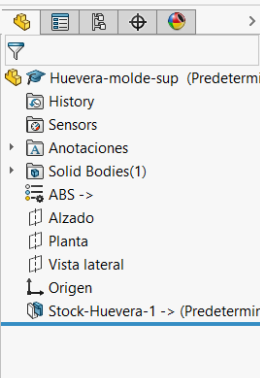
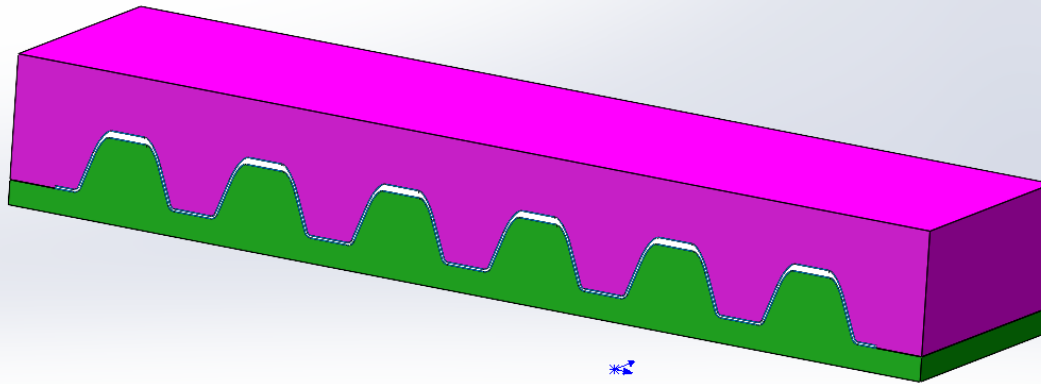
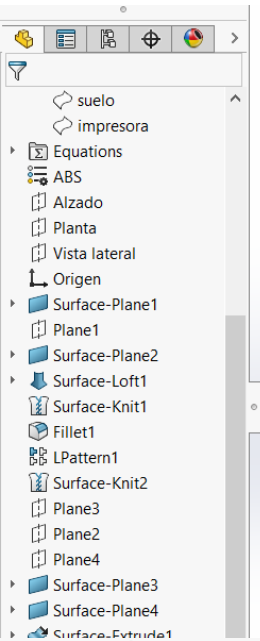
Impact & fracture properties

Fracture toughness	* 4	- 5	MPa.m <sup>0.5</sup>
--------------------	-----	-----	----------------------

Thermal properties

Glass temperature	47	- 67	°C
Maximum service temperature	77	- 130	°C
Minimum service temperature	-273		°C
Thermal conductivity	0.1	- 0.15	W/m.°C
Specific heat capacity	1.3e3	- 1.4e3	J/kg.°C
Thermal expansion coefficient	5	- 20	µstrain/°C

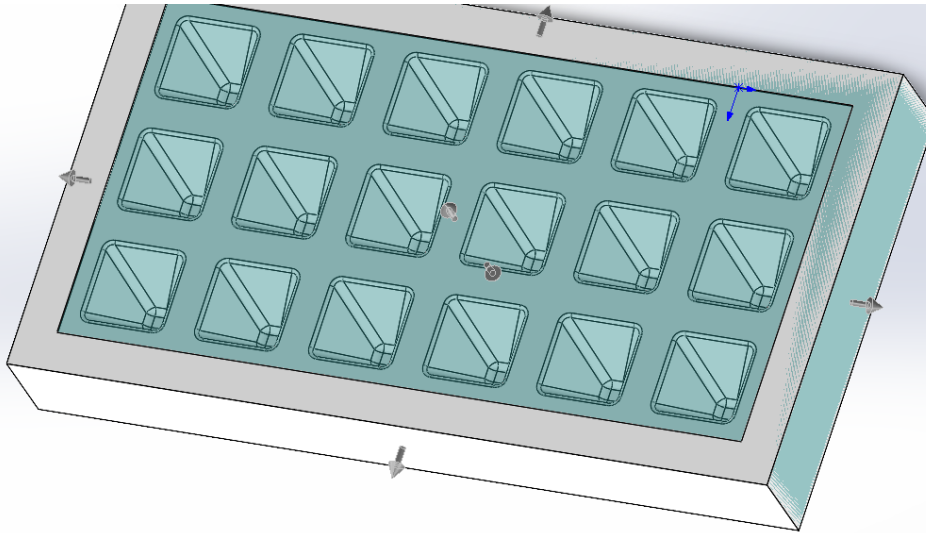
# • Moldes y costes





## • Moldes y costes

- Setup (2) [2.80 USD]
- Mill Operations (1) [0.39 USD]
- Custom Operations [0.00 USD]
- ⚠ No Cost Assigned (18)



Launch Template Editor...

**Material**

Class: Steel

Name: AISI 304

Material cost: 9.80 USD/lb

Weight: 77.60 lb

**Stock Body**

Type: Block

**Estimated Cost Per Part**

763.69 USD/Part

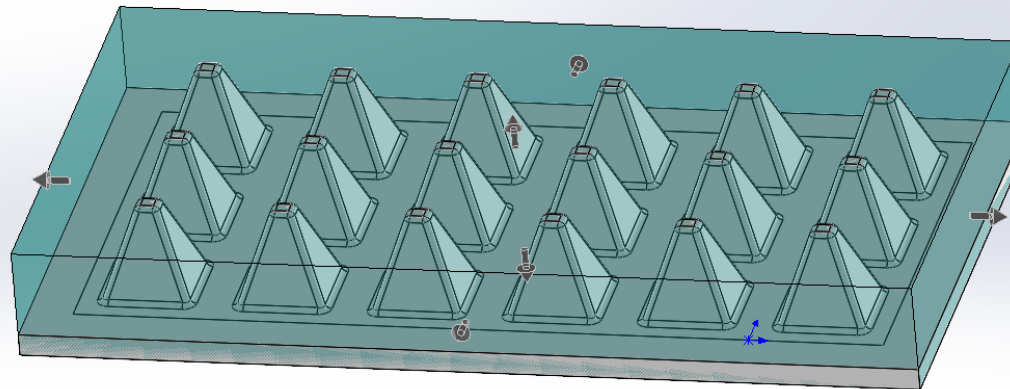
Comparison 100%

Current **763.69 USD**

Previous 0.00 USD

Breakdown

- Setup (2) [2.80 USD]
- Mill Operations (1) [11.65 USD]
- Custom Operations [0.00 USD]
- No Cost Assigned



Launch Template Editor...

**Material**

Class: Steel

Name: AISI 304

Material cost: 9.80 USD/lb

Weight: 77.60 lb

**Stock Body**

Type: Block

**Estimated Cost Per Part**

774.95 USD/Part

Comparison 100%

Current **774.95 USD**

Previous 0.00 USD

Breakdown

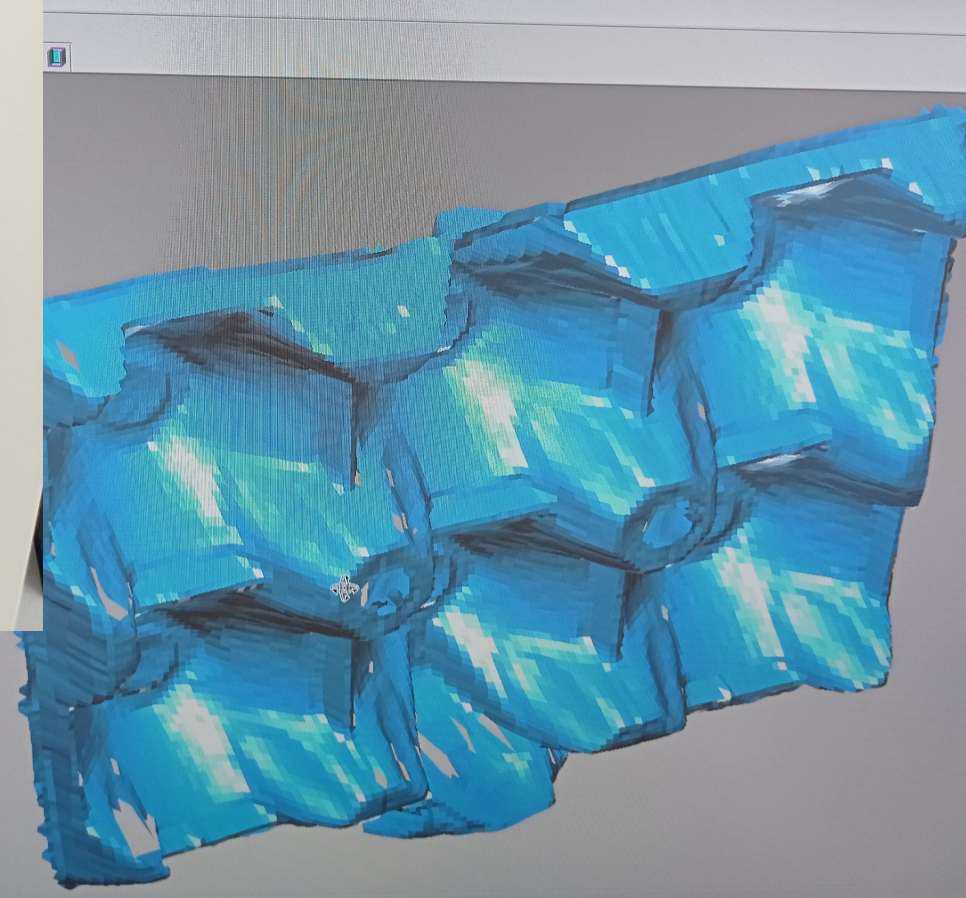
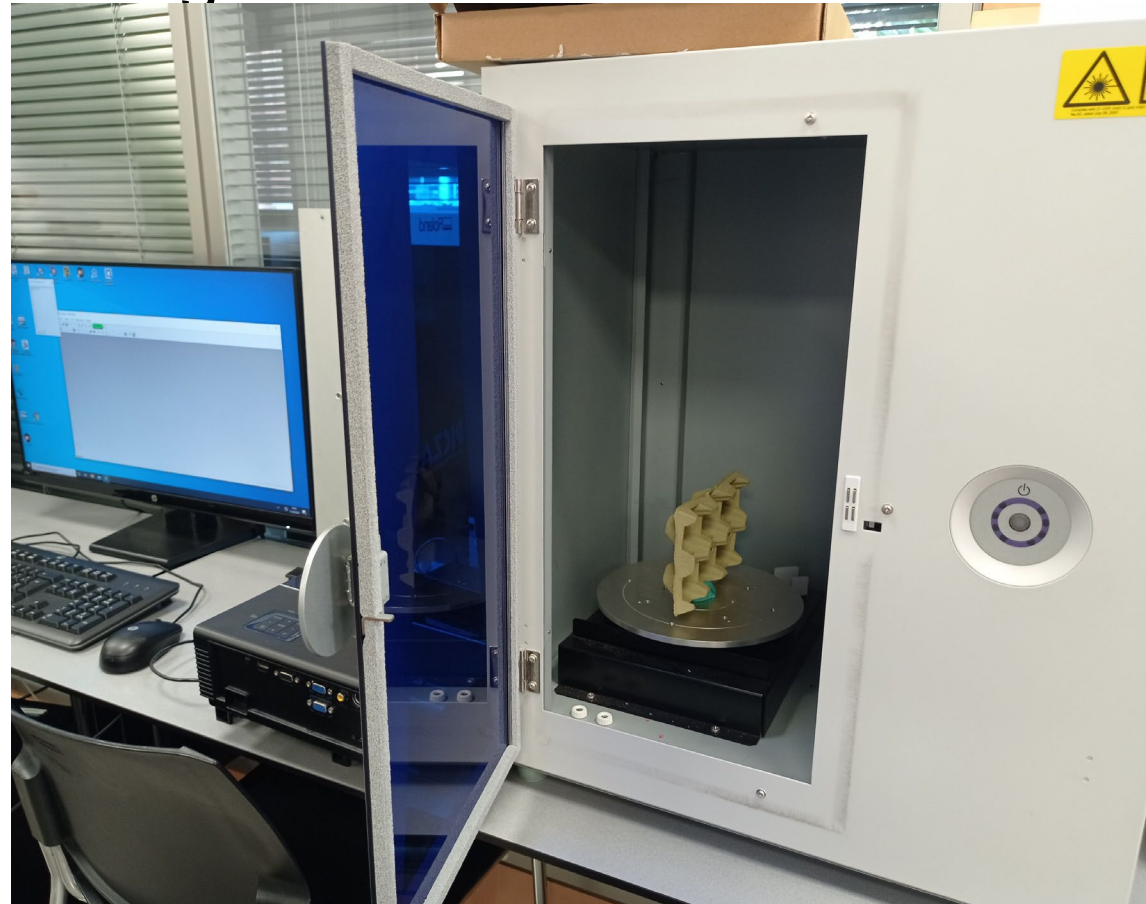
## • Ensayos experimentales

Como no tenemos geometría real se va a mirar algo similar usando hueveras comerciales del supermercado. Tomando cuatro muestras de peso vemos una gran variación de 10.70 a 12.29 gramos que indica el poco control de esta técnica.

			sample1	sample2	sample3	sample4	media	max-med	med-min	max-min
medición	peso	gr	11.52	10.70	12.29	11.65	11.54	6.50%	7.28%	14.86%

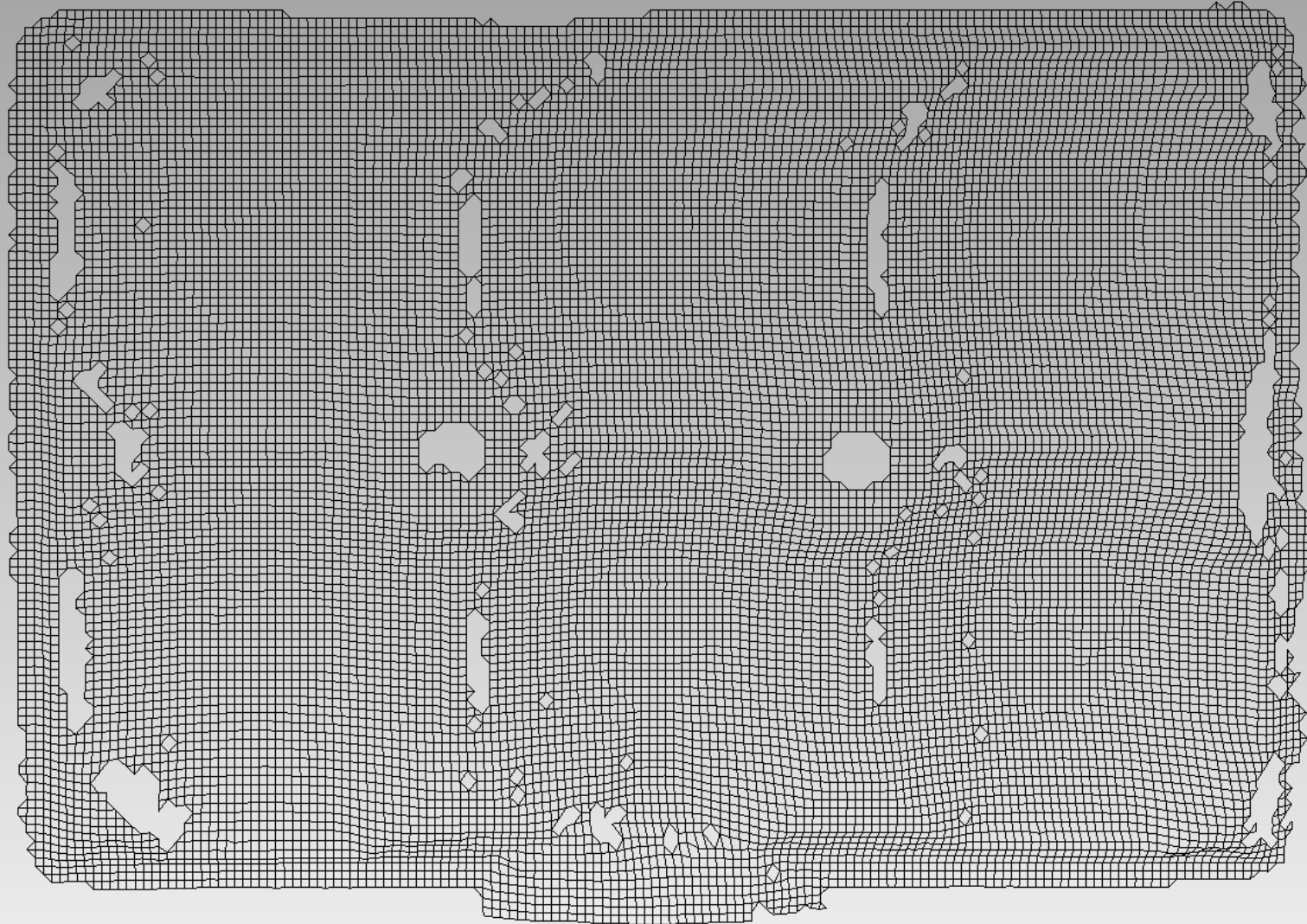
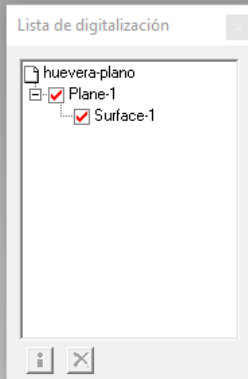


- Digitalizado

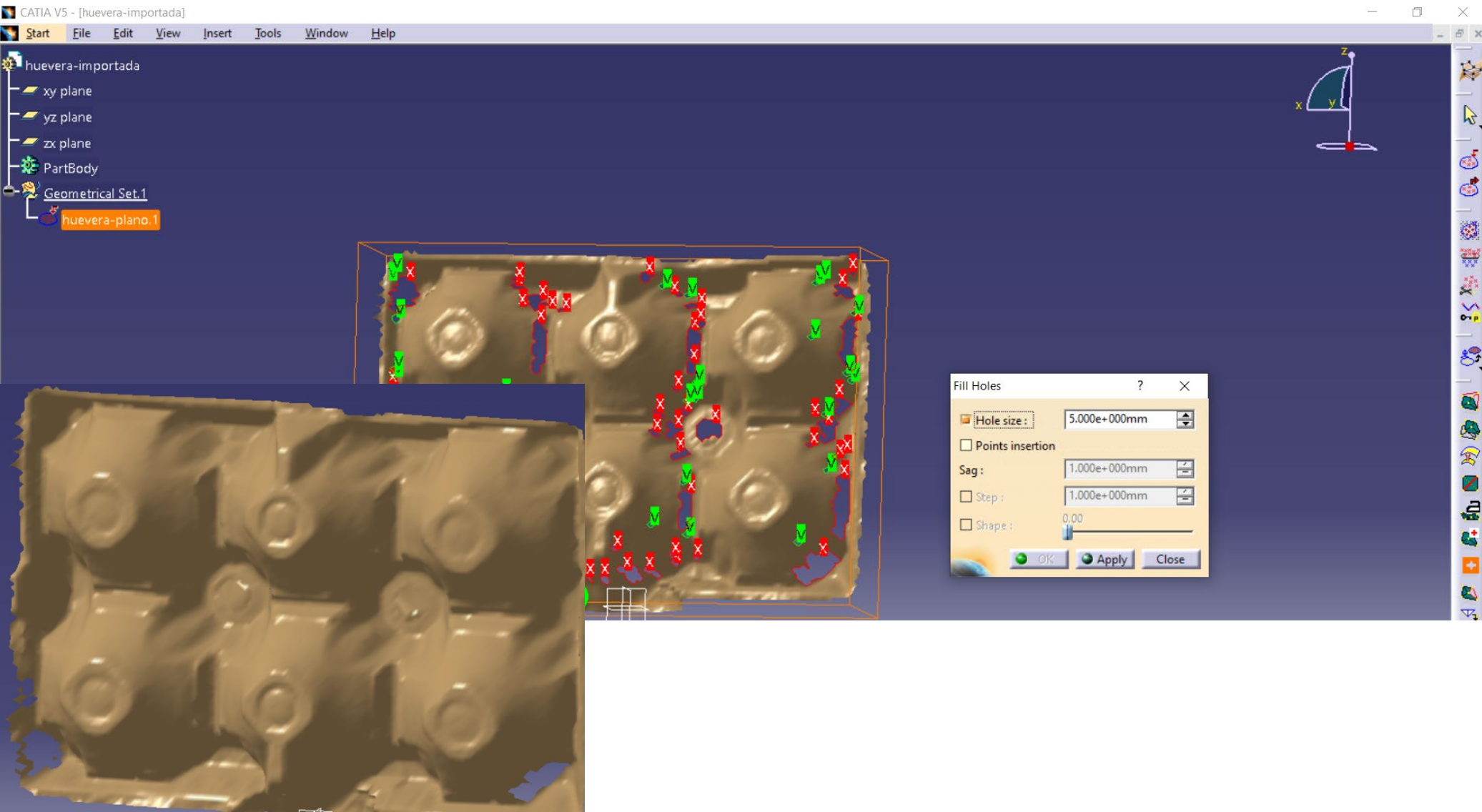


huevera-plano.pij - Dr.PICZA3

Archivo Editar Ver Seleccionar Ayuda



# • Arreglamos el mallado en Catia



# • Arreglamos el mallado en ANSA

ANSYS ANSA v21.1.0 64-bit

File Tools Utilities Lists Assembly Plugins Windows Help

Search Functions and Filters

New Ctrl+N

Open Ctrl+O

Open recent

Merge

Input

Input recent

Output

Save

Save as Ctrl+S

Save as v20

Save visible as

Save All

Output CAD

Input connections

Output connections

Input model definition

Output model definition

Input session file

Release License

Quit Ctrl+Q

Recent files

topology.nas //sdoc1/software/Exp\_Grafica/...NASTRAN\_SOL200/tutorial\_files

Topography\_final.nas //sdoc1/software/Exp\_Grafica/...NASTRAN\_SOL200/tutorial\_files

al.ansa //sdoc1/software/Exp\_Grafica/...NASTRAN\_SOL200/tutorial\_files

final.ansa //sdoc1/software/Exp\_Grafica/...NASTRAN\_SOL200/tutorial\_files

nal.ansa //sdoc1/software/Exp\_Grafica/A...nsa\_fluent\_meta/tutorial\_files

PLOT3D

SU2

MEDINA

Tosca Structure

ONF

PATRAN

I-DEAS

SONATE

FREE FORM

OFF FORM

SHL FORM

BETA TANK

WAVE FRONT

INVENTOR

ST-LITHGRP

VRML

FiberSim metadata

ADAMS

MotionSolve

ANSA Thickness

MOLDFLOW

Drop files here to open

ANSYS PRE PROCESSOR

Modules Buttons

Hot Points

Insert	Project	Parametri
Delete	Mult.Proje	Intersect
Release	Weld Spot	

CONS

Paste	Release	Fill Hole
Open Hole	Project	Break

Faces

Cut	Delete	Undelete
New	Set PID	Topo
Proj.Cut	Mid.Surf	Rm.Log
Fuse	Flange	Rm.Dbl
Plane C	Offset	Freeze/Un
Zone Cut	Convert	Dach
Intersect	Extend	Orient

Modify

Surfaces

Coons	Fit	Extend
Volume	Extrude	Break

Contoured

Curves

Create	Delete	Undelete
Connect	Transform	Imprint

Points

New	Delete	Undelete
Relative	On COG	On Curve

Auxiliaries

Working	Cross S	Fine
IGA		

Options List

NOTICE: script [C:/Users/andres.garcia/.BETA/ANSA/versio  
NOTICE: script [C:/Users/andres.garcia/ANSA\_TRANSL.py]

# • Arreglamos el mallado en ANSA

ANSYS v21.1.0 64-bit (C:/Users/andres.garcia/OneDrive - IQS/ASIGNATURAS/ANSYS/huevera-plano.stl,ans)

File Tools Utilities Lists Assembly Plugins Windows Help

Search Functions and Filters

Database

Name	Number	Visible
ANSAPART 2		
EDGE		
<input checked="" type="checkbox"/> > ELEMENT	31033	31033
<input type="checkbox"/> > GEOMETRY 3	0	
<input type="checkbox"/> GRID	16096	
> MATERIAL 1		
> PROPERTY 1		

0:huevera-plano.stl, Current Part: huevera-plano

Shell  
quads : 0  
trias : 31033  
total : 31033

Modules Buttons

**Hot Points** Info

Insert Project Parametri.  
Delete Mult.Proje. Mark/Un

**Perimeters** Info

Number Num +/- Spacing  
Init Length Align  
FE Peri

**Macros** Info

Cut Release Orient  
Join Proj.Cut Edge2Peri.  
Freeze/Un Set PID Simplify

**Grids** Info

Move Origin Align  
Paste New Thickne.

**Mesh Generation**

Free Remesh Batch  
Spot Me Erase STL  
Adv.Fro. 4 Sided Circular

**Shell Mesh**

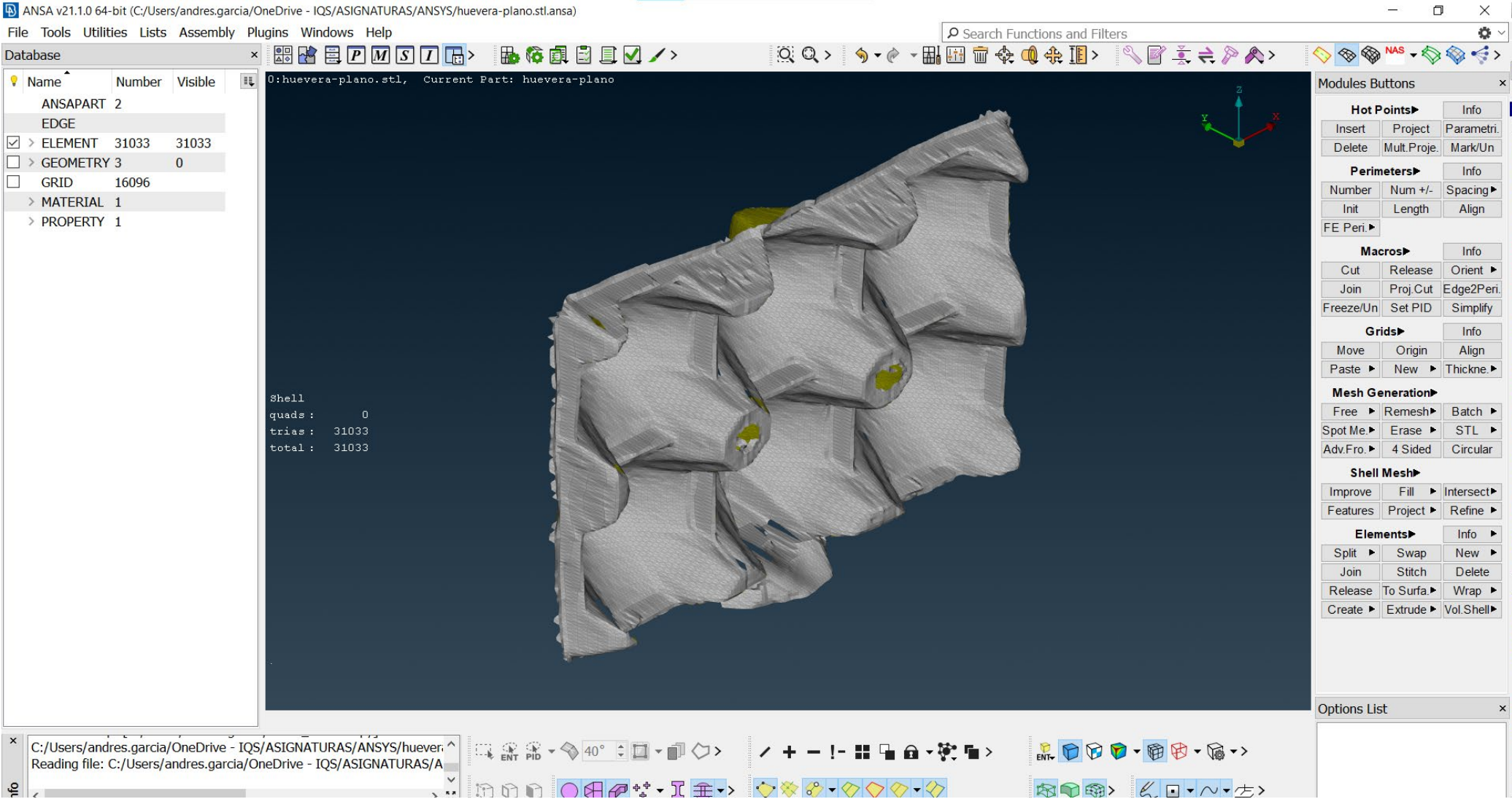
Improve Fill Intersect  
Features Project Refine

**Elements** Info

Split Swap New  
Join Stitch Delete  
Release To Surfa Wrap  
Create Extrude Vol.Shell

Options List

C:/Users/andres.garcia/OneDrive - IQS/ASIGNATURAS/ANSYS/huevera-plano.stl  
Reading file: C:/Users/andres.garcia/OneDrive - IQS/ASIGNATURAS/A





# • Arreglamos el mallado en ANSA

ANSYS v21.1.0 64-bit (C:/Users/andres.garcia/OneDrive - IQS/ASIGNATURAS/ANSYS/huevera-plano.stl.ansa)

File Tools Utilities Lists Assembly Plugins Windows Help

Database

Name	Number	Visible
ANSAPART 2		
EDGE		
<input checked="" type="checkbox"/> > ELEMENT	31033	31033
<input type="checkbox"/> > GEOMETRY 3	0	
<input type="checkbox"/> GRID	16096	
> MATERIAL 1		
> PROPERTY 1		

0:huevera-plano.stl, Current Part: huevera-plano

Shell  
quads : 0  
trias : 31033  
total : 31033

Fill>Holes Select EDGE

Reading file: C:/Users/andres.garcia/OneDrive - IQS/ASIGNATURAS/A  
Identified 86 hole(s).

Holes

Options Set identification options

Preview

Fill mode Single bound holes

FE fill method Draft

Geom fill method Create new/extend e

Max diameter 50.

Identify external perimeters

Identify

Next > Cancel

Modules Buttons

Hot Points Info

Insert Project Parametri.  
Delete Mult.Proje. Mark/Un

Perimeters Info

Number Num +/- Spacing  
Init Length Align  
FE Peri

Macros Info

Cut Release Orient  
Join Proj.Cut Edge2Peri.  
Freeze/Un Set PID Simplify

Grids Info

Move Origin Align  
Paste New Thickne.

Mesh Generation

Mixed.. CFD 2nd Ord  
Reconst Best Gradual  
Map

Improve Fill Intersect  
Features Project Refine

Elements Info

Split Swap New  
Join Stitch Delete  
Release To Surfa Wrap  
Create Extrude Vol.Shell

Options List

Element type	mixed
Target length	averag
Defeaturing le...	0.667*
<input type="checkbox"/> Minimum le...	0.

# • Arreglamos el mallado en ANSA

ANSYS v21.1.0 64-bit (C:/Users/andres.garcia/OneDrive - IQS/ASIGNATURAS/ANSYS/huevera-plano.stl.ansa)

File Tools Utilities Lists Assembly Plugins Windows Help

Database

Name	Number	Visible
ANSAPART 2		
EDGE		
<input checked="" type="checkbox"/> > ELEMENT	31638	31638
<input type="checkbox"/> > GEOMETRY 3	0	
<input type="checkbox"/> GRID	16096	
> MATERIAL 1		
> PROPERTY 1		

0:huevera-plano.stl, Current Part: huevera-plano

Shell  
quads : 0  
trias : 31638  
total : 31638

Delete Select SHELL

MODEL ID: 0.  
Identified 1 hole(s).

Modules Buttons

Hot Points ▶ Info

Insert Project Parametri.  
Delete Mult.Proje. Mark/Un

Perimeters ▶ Info

Number Num +/- Spacing ▶  
Init Length Align

FE Peri. ▶

Macros ▶ Info

Cut Release Orient ▶  
Join Proj.Cut Edge2Peri.  
Freeze/Un Set PID Simplify

Grids ▶ Info

Move Origin Align  
Paste ▶ New ▶ Thickne. ▶

Mesh Generation ▼

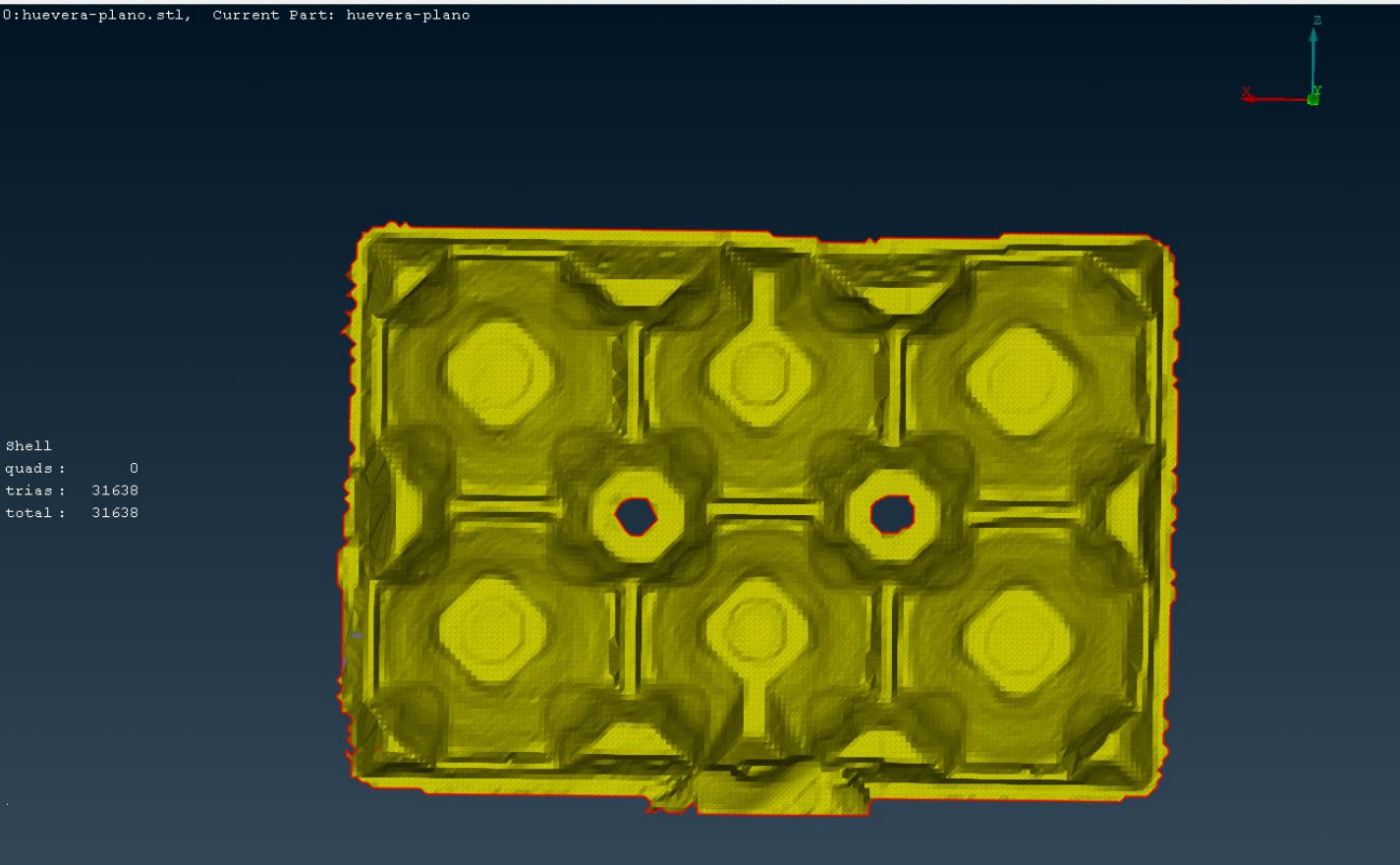
Mixed.. ▶ CFD ▶ 2nd Ord  
Reconst. Best ▶ Gradual ▶  
Map ▶

Improve Fill ▶ Intersect ▶  
Features Project ▶ Refine ▶

Elements ▶ Info ▶

Split ▶ Swap New ▶  
Join Stitch Delete  
Release To Surfa ▶ Wrap ▶  
Create ▶ Extrude ▶ Vol.Shell ▶

Options List



# • Ensayos huevera compresión

Primero ensayamos la huevera a compresión para obtener la curva fuerza-desplazamiento y obtener la energía.



Handwritten notes on a whiteboard:

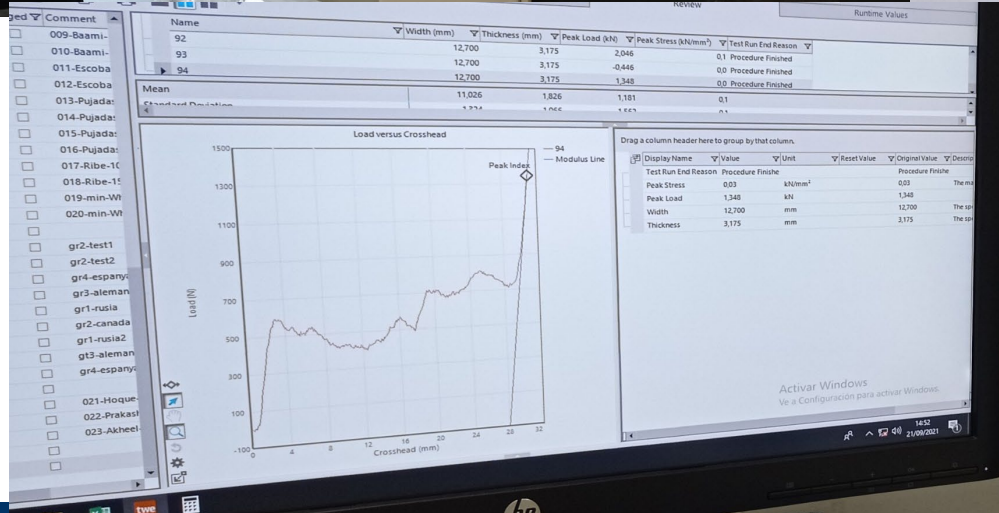
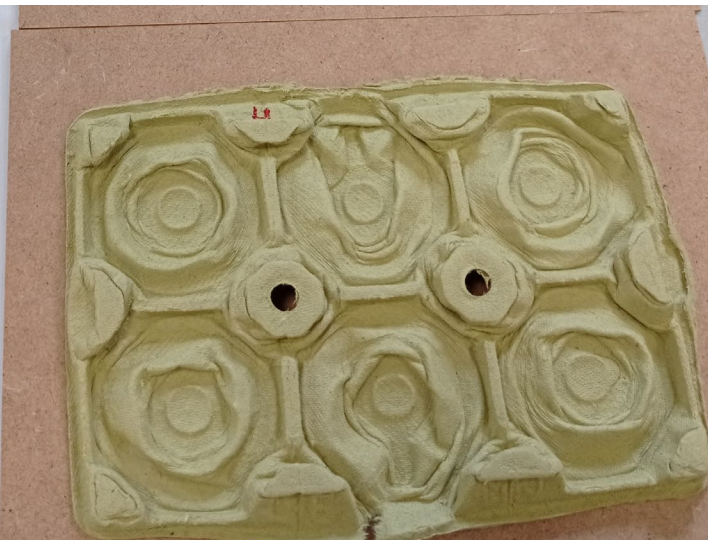
$$E = \int F dx \sim 10^3$$

$$V = \sqrt{2E/m} \quad \sqrt{\frac{2 \cdot 10}{0.36}} = 76 \text{ m/s}$$

$$h = \frac{E}{mg}$$

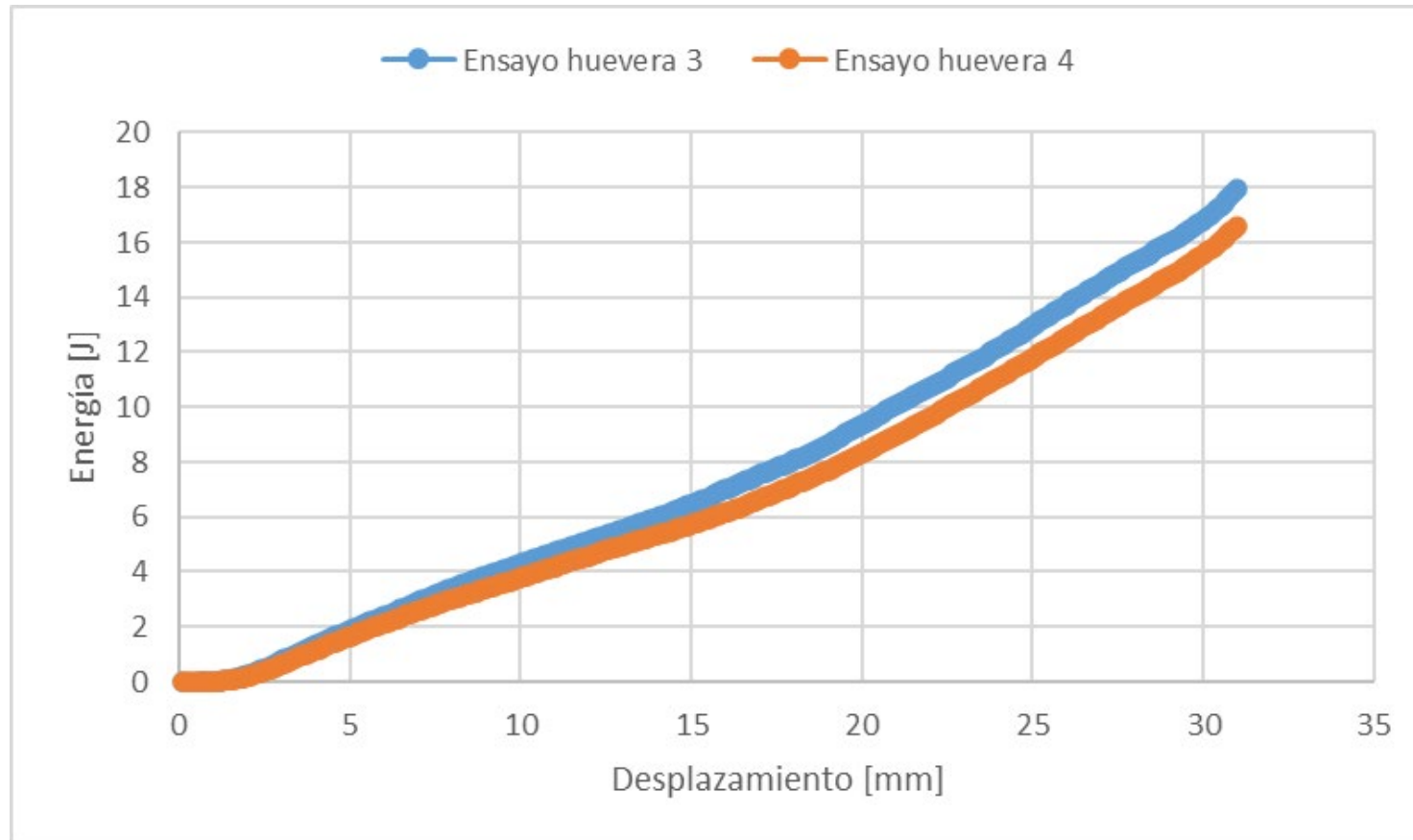
$$a = \frac{F_{max}}{m}$$

$$t = \frac{V}{a}$$



## • Ensayos huevera compresión

Una vez tenemos el ensayo podemos ver como hacer el impacto



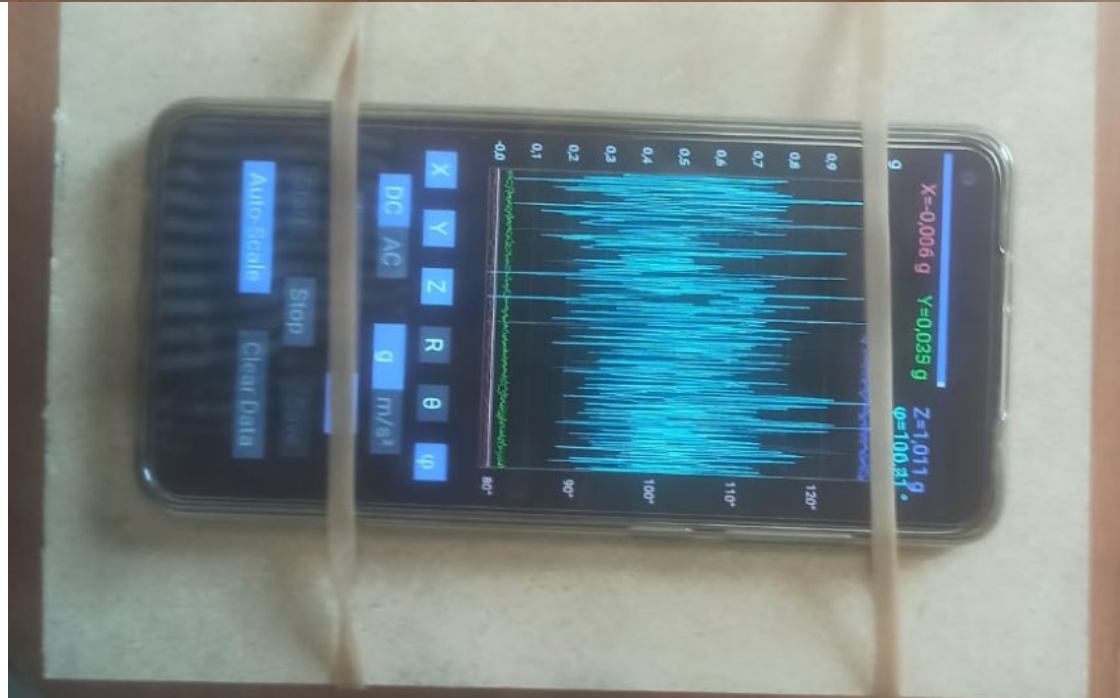
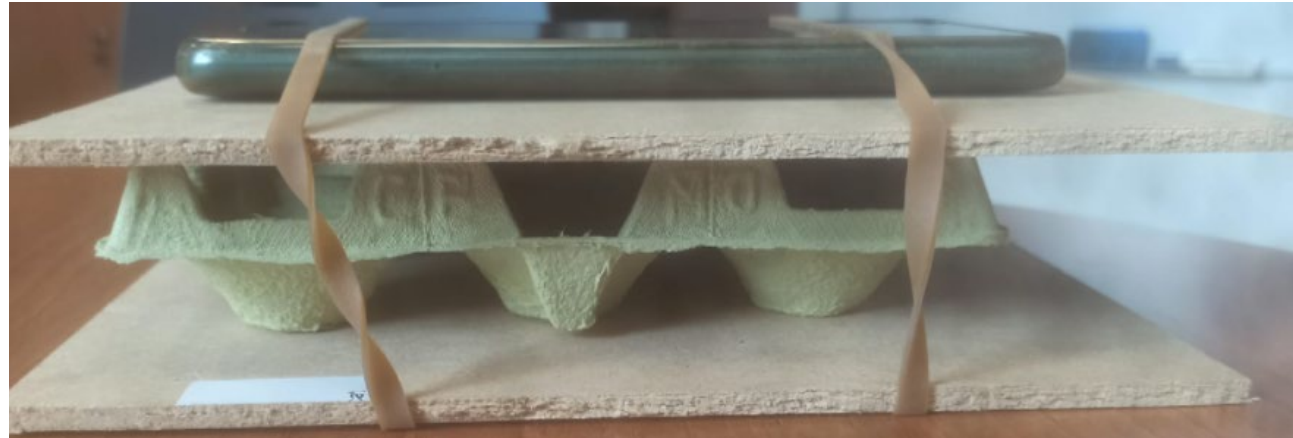
## • Ensayos huevera impacto

La masa de cada madera es de 105g y el móvil 230g gomas 1.5 g total 443g. La masa que comprime es de 105+230+1=346g aprox.

Para tener la energía despejamos la velocidad:  
 $v = \sqrt{(2E/m)}$ , altura  $h = E/(mg)$  y la aceleración máxima esperamos esté entorno a:  $a_{\max} = F_{\max}/m$

El tiempo lo podemos estimar como  $t = v/a$

El móvil lee cada 2.5ms por lo que tenemos difícil ver bien el impacto y tan sólo  $\pm 15[g]$ .



## • Resumen.

- Automatización de cálculos variando espesor o propiedad de material.
- Revisar los cálculos automatizados para no dar un valor con simulación que no ha acabado o problema de energías
- Posibilidad de tener la optimización parametrizada con la geometría CAD bien relacionada.
- Conclusiones de modelo sencillo parecidas al modelo completo de huevera.
- Posibilidad de creación de scripts
- Base de materiales GRANTA
- Costes de moldes
- Ensayos de PaperPulp con hueveras con grandes diferencias en peso
- Escaneo de piezas para hacer ingeniería inversa

## • Resumen CURSO.

- Cálculo a tracción para verificar propiedades de material y saber que hacemos bien energías, deformaciones, tensiones, fuerzas...
- Impacto para ver como la energía cinética se transforma en energía interna con un pequeño rebote elástico.
- Impactos a flexión para poder verificar impactos de manera sencilla.
- Importancia del Time step. Par acero de 10[mm] el time step son unos 2[us] por lo que para calcular 20[ms] se necesitan 10000 pasos de cálculo. Si tenemos un elemento de 1[mm] no nos podemos permitir hacer 100000 pasos de cálculo y es importante hacer bien el mass scaling.
- Importancia de automatizar para poder lanzar muchos cálculos y evaluar riesgos de si es más duro o más blando.