# RESEARCH PROJECT:
# SHOCK WAVES PROPAGATION ON MATERIALS. APPLICATION TO LASER SHOCK MODELLING IN ABAQUS

Author
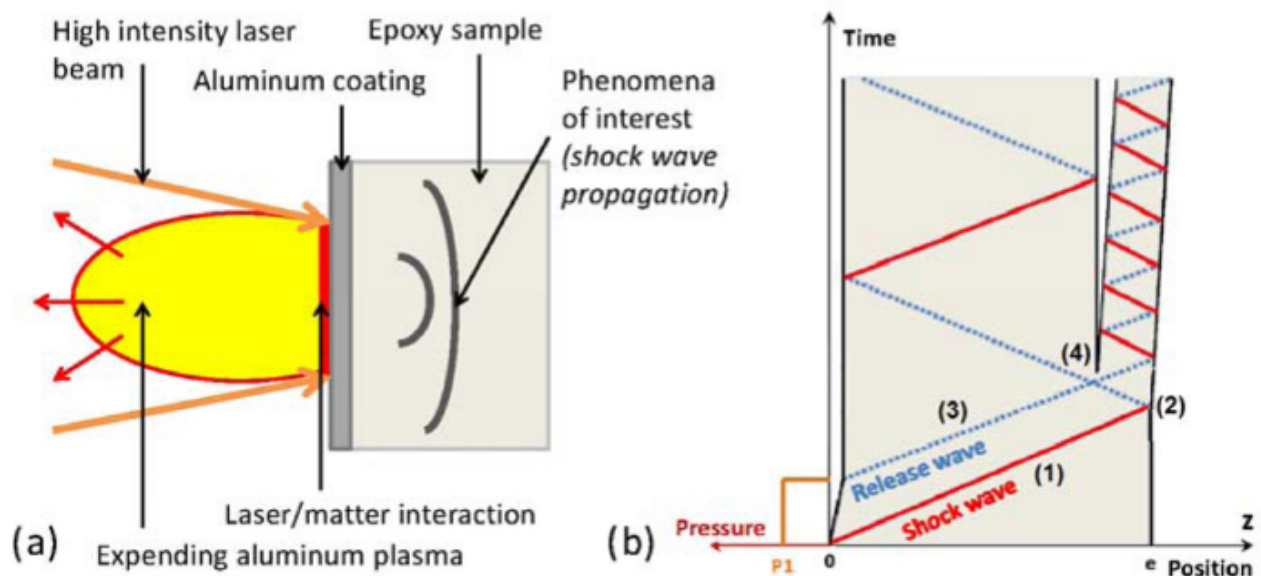
## Eusebio Palmero Buendía

**eusebio.palmero-buendia@etu.isae-ensma.fr**

Academic condutors

## Marianne Beringhier    Michel Boustie

marianne.beringhier@ensma.fr    michel.boustie@ensma.fr

**MSc in Aeronautical Mechanics and Energetics**

Major in Structures-Materials

**ISAE-ENSMA**

**École Nationale Supérieure de Mécanique et d'Aérotechnique**

Futuroscope Chasseneuil Cedex (France) - June 2017

# Contents

# 1   Introduction

The formation of shock waves by the incidence of laser is a technique which is continuously growing nowadays due to its several applications. Testing with laser consists on applying a certain effort induced by high-powered laser beam over a surface within a brief interval of time. The laser light has its specific features, such as a certain wavelength (whatever except white light), way of propagation (precise direction along the surface) or magnitude (strong concentration of energy during the time of application).

We can distinguish different laser shock techniques or testing types depending of the some specific parameters: wavelength, application field, regime of applicantion (continuous or impulsive), the power or energy per impulsion, frequency, impulsion durability... Among the principal effects of the laser application, there can be the dynamic damage on materials, solid behaviour laws, adherence phenomena or dynamic compaction.

Improving fatigue or corrosion behaviour of metallic materials through the generation of compressive stress field is one of the features of laser shock processing. In addition, the numerical prediction or morphological and mechanical surface modifications can be adressed by the analysis of shock wave propagation.

The first section of this report contains a sum up of the mechanics of shock waves, which includes the base equations (conservation equations), characteristic curves and way of propagation. Our aim is creating a numerical modelling by a Finite Element Method software, Abaqus, in order to compare with the analytical results taken from the laser shock propagation theory. This can allow us to validate mechanical loading and materials' constitutive laws.

The procedure along the whole document will cover the beginning with a simulation of an impact test. In addtion, we will approach the application of a distribution of pressure depending on time over a surface what represents the nature of the laser beam effort. The idea is modelling geometries composed by both different isotropic materials with a future perspective of the combination of them with composite materials (anisotropic).

# 2   Theoretical brackground

Generally, a shock wave can be defined as the propagation of a pressure discontinuity during a brief interval of time or along a reduced distance of the order of a few nanometers. It carries energy and must be propagated through a medium.

A shock wave is characterised by an abrupt change in state variables (pressure, temperature, internal energy and density or specific volume), what define the themodynamic state of the medium. It passes from an initial condition $(P_0, T_0, E_0, \rho_0)$ to a final state $(P_1, T_1, E_1, \rho_1)$ divided by the shock wave. The energy of a shock wave dissipates relatively quickly with distance or time.

Considering shock waves with high amplitude discontinuities, the corresponding material which we are working with is considered as a fluid. Therefore, its behaviour under the pass of a shock wave is defined by the three hydrodynamic laws of conservation (mass, energy and momentum) [1]. Two different velocities have to be distinguished in these equations: the propagation velocity of the wave "$D$" and the velocity of the material "$u$" induced in the medium.

$$\rho_0(D_{01} - u_0) = \rho_1(D_{01} - u_1) \tag{1}$$

(mass/continuity equation)

$$E_1 - E_0 = \frac{1}{2}(P_1 + P_0)(\frac{1}{\rho_0} - \frac{1}{\rho_1}) \tag{2}$$

(energy)

$$P_1 - P_0 = \rho_0(D_{01} - u_0)(u_1 - u_0) \tag{3}$$

$$\text{(momentum)}$$

Appart from the three laws, we need another expressions which complement the previous ones. These are the equations of state [1] of the different materials used in our case, which help to determine the state of the material with the fluid notion:

$$f(P, E, \rho) = 0 \tag{4}$$

However, it seems to be necessary to combine the hydrodynamic mechanics that the material acquires when it is submitted to a high deformation rates and considered a fluid (our assumptions for laser shock modelling), with the elastoplastic notions associated to solids. The elastoplastic behaviour is defined by the characteristic elastic limit of Hugoniot [1].

$$P_H = 1 + \frac{\lambda}{2\mu}\sigma_{Y0} \tag{5}$$

where $\lambda$ and $\mu$ are the Lamé's coefficients and $\sigma_{Y0}$ is the elastic limit of the material.

This limit separates two ways of wave propagation, i.e from a certain value of pressure $P_H$ the waves stop being propagated under an elastic form ($P < P_H$) with velocity $C_e$ and are propagated under a combination of both elastic and plastic waves ($P > P_H$) with a different velocity or $C_p$.

The shock propagation velocity is given by a linear combination of velocities, such as the sound velocity in the corresponding material "$c_0$" plus the Hugoniot's constant "$s$" times the induced velocity: $D = C_0 + su$. Then, the expression of the induced pressure, what gives its theoretical hyperbolic shape of the shock polar curve (or shock Hugoniot's curve), is [1]:

$$P = \rho C_e(C_0 + Su) \tag{6}$$

The characteristic curves of the material under the action of the shock wave are composed by three different regimes. They allow to stablish the relation between the atmospheric conditions and define the propagation regimes of the waves. These curves are the Hugoniot curve, relation of pressure and specific volume "$V$", and the shock polar curve, definition of the pressure as a function of material velocity $P = f(u)$. Both curves start from an initial state of pressure at ambient conditions "$P_0$" and the definition (6) is valid for both elastoplastic and hydrodynamic regimes.

Hugoniot curve (dynamic adiabatic) can be directly interpreted in terms of the pressure wave propagation in the corresponding medium from an initial state at ambient pressure. It has three regions limited by the Hugoniot limit: an elastic part where the velocity does not depend on on the shock pressure under the limit HEL in the picture, an elastoplastic region defined by the intersection of the Rayleigh's line, prolongated from the elastic part, with the Hugoniot curve and finally a hydrodynamic part above the Rayleigh's line where the elastoplastic behaviour of the material can be neglected:
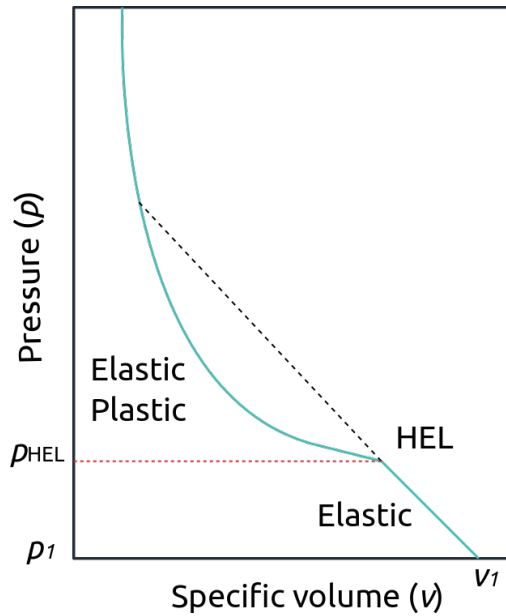
Figure 1: Example of Hugoniot's curve

The shock polar curve allows to characterize the state of the material with the two main accessible variables, pressure and velocity (P,u), under shock from its initial state. Normally, we are going to work with the linear expression of the induced pressure previously defined, valid both in hydrodynamic and elasto-plastic regimes, which allows to build the curves easily. It is composed by three different regions like the Hugoniot curve, limited by the elastoplastic behaviour of the material.
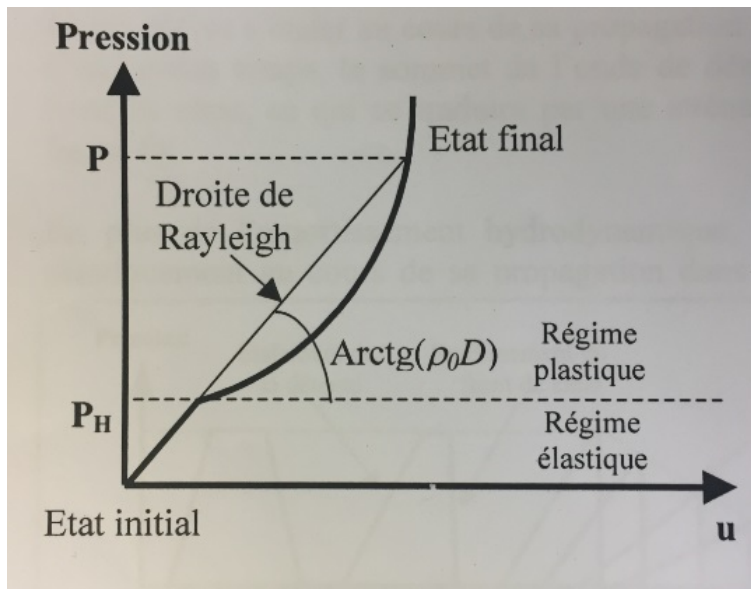


Figure 2: Example of shock polar curve [1]

The characterization of shock waves requires a huge database in order to reproduce the previous curves for a great variety of materials. At least two parameters of each equation of state (sound velocity, density...) need to be simultaneously measured from the experience. It allows to define the relative slope of the polar curves which clasifies the material response to shock according to its impendance ($Z = \rho D$). The higher the impedance of the material, the harder its deformation results. In our case, we are going to used the literature to perform our analysis.

Finally, the main assumption of laser shock modelling is considering only the hydro-dynamic behaviour of the material, i.e the region of the previous curves where they are considered fluids. In that regime, the deviatory part $\sigma_d$ (deformation term) of the applied stress is negligible compared with the spherical part (pressure times inertia term) which corresponds to the hydronamic term, from the following definition [1]:

$$\sigma = -pI + \sigma_d \tag{7}$$

In addition, it is assumed that the induced pressure follows the linear expression $P = \rho C_e u$ in this regime, what modifies the hypotetical Hugoniot's curves.

5

# 3 Abaqus modelling

## 3.1 Preliminary settings

Abaqus is a Finite Element Method software which can approach many kind of problems due to the possibility to choose different solvers. Our aim is proving the applicability of such quasi-static numerical method to abord the laser shock modelling.

This section approaches the elementary commands that have to be set inside the software in order to reproduce a good model of our simulations. First of all, the selection of the solver in Abaqus has to be set to explicit/standard instead of implicit, which implies that the results can be shown in every simulation step instant depending on the previous step. It means that an initial step is defined by default and the time and space increments need to be controlle due to the stability in the problem resolution (convergence phenomena).

The time increment plays an important role in a dynamic phenomena, like the propagation of a discontinuity, to get a good behaviour in the computation. It should be enough small to ensure the problem stability and convergence in order to display accurate results of the case. Between the advantages of the explicit solver, there are the precision in the results conditioned by the small time step, the moderated computational cost and the applicability to non-linear and dynamic problems.

We should define a stability criteria relating both increments with a specific variable of the problem, which can come from an experimental test. This criteria needs to comply with the convergence condition: every perturbation cannot pass a mesh element size (space increment) in more than one time increment. Normally, it is going to be imposed for us because we are not able to perform such experimental part.

Working in Abaqus also requires to be consistent with the units for every variable defined during the modelling part. Here we have a table for the equivalences depending on the units of length chosen when defining the geometry:

| Base units | | | Derivated units | | | | |
|---|---|---|---|---|---|---|---|
| **Lenght** | **Time** | **Mass** | **Force** | **Pression** | **Energie** | **Velocity** | **Density** |
| $[L]$ | $[T]$ | $[M]$ | $[M][L][T]^{-2}$ | $[M][L]^{-1}[T]^{-2}$ | $[M][L]^{2}[T]^{-2}$ | $[L][T]^{-2}$ | $[M][L]^{-3}$ |
| M (SI) | S (SI) | Kg (SI) | N | Pa | J | $m/s$ | $kg/m^3$ |
| mm | ms | g | N | MPa | mJ | $m/s$ | $kg/cm^3$ |
| cm | $\mu s$ | g | MdaN | Mbar | $10^5 J$ | $cm/\mu s$ | $g/cm^3$ |
| $\mu m$ | ns | pg | mN | GPa | nJ | $km/s$ | $g/cm^3$ |

Table 1: Equivalence in units [5]

## 3.2 Projectile impact over a metallic plate

In this test, we are going to simulate an impact test of a tantalium/aluminum made impactor and the objective tantalium plate. The impact creates a shock wave which is propagated along the objective plate with its corresponding reflections due to change of interfaces.

Such a case will be analytically calculated according to shock waves physics in order to perform a direct comparison between theory and simulation. Doing so, we can ensure the validity of many choices to be made by the engineer when setting the simulation.

The aim is to charaterize dynamically the materials which are submitted to extreme constraint conditions and deformations, with the spalling phenomena taking place. That induces the damage because of tensile stresses by the reflection in the back face.
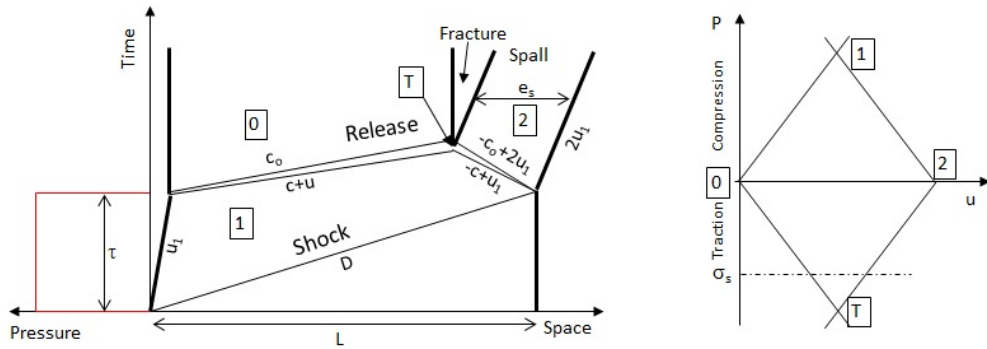
Figure 3: Example of shock wave propagation with spalling

The principle of spalling consists on reproducing the solicitations in traction which leads to the material damage by shock propagation. The briefness and the strong amplitude of the laser shocks allow to study the fracture of the materials subjected to high deformation rates (order of $10^7$ $s^{-1}$). It is possible to study the sample after the test due to its non-destructive nature and the data extraction permits to explore a new domain. The effects of material damage are observable from the failure data associated with a corresponding back face velocity previously categorized with the tensile stress value.

The following steps of the report include the procedure performed for the definition of Finite Element model in Abaqus (geometry, material properties, time step definition, load/predefined field and boundary conditions, interaction properties, mesh, output fields...).

### 3.2.1 Pre-processing and Abaqus Running

- **Geometry:**
  The axisymmetry of the cylindrical sample allows to simplify the geometry in a cuadrangular flat section (0.8x0.8 cm) as well as for the projectile, formed by the upper impactor of tantalium (0.5x0.1 cm) and the down impactor of aluminum (0.5x0.2 cm). The simplifications in the geomety from symmetry allow to consider a 2D problem:
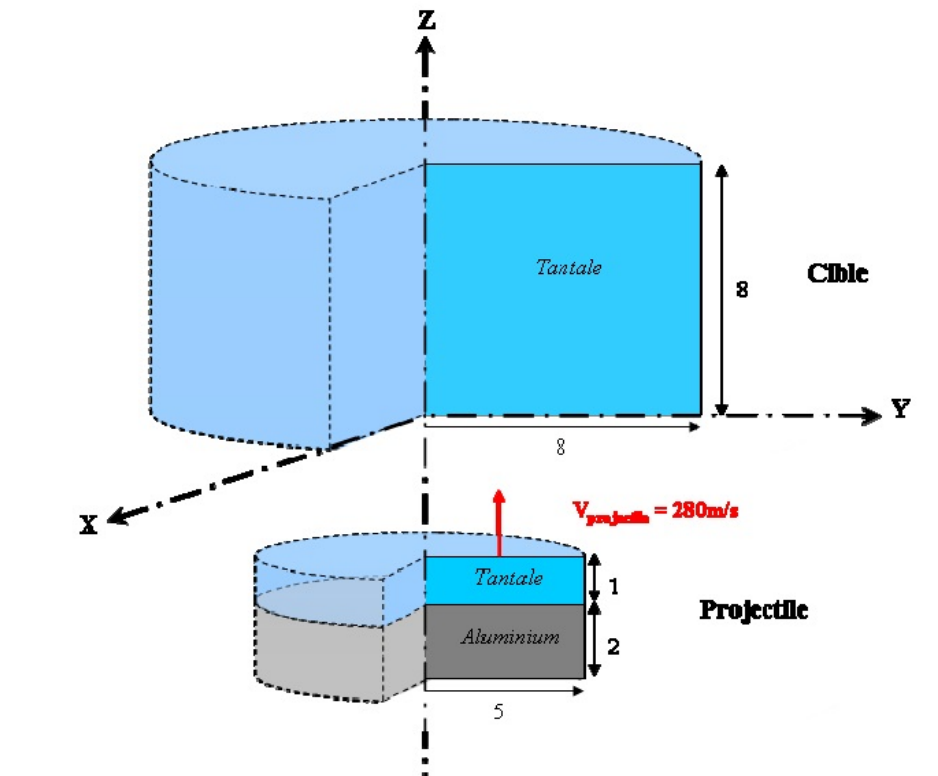


Figure 4: Geometry of impact test case [5]

The different parts of the problem are defined in the module "Create Part" selecting Modelling Space "Axisymmetric" (Y-axis by default in Abaqus), Type "Deformable" and Shape "Shell". The objective plate has its lower left vertex at the origin of coordinates and is separated 0.1 cm from the projectile, whose upper left vertex is at (0,-0.1).

Since the impact is produced in only one direction, the problem can be treated as 1 dimensional one due to the sense of the shock wave propagation. However, 2D effect are going to be induced as well as it advances after the impact.

- **Materials:**
The next step is defining the materials Aluminum and Tantalium in the Property module. Between the properties to be defined, we find the density inside the General menu and the parameters of the equation of state (EoS) characteristic of each one inside the Mechanical menu, type Us-Up. The following table contains the values of every variable which must be transformed in the right units:

| Material | $\rho_0(g/cm^3)$ | $c_0(cm/\mu s)$ | $s(-)$ | $\gamma(-)$ |
|---|---|---|---|---|
| Aluminum (Al) | 2.785 | 5.328 | 1.338 | 2.09 |
| Tantalium (Ta) | 16.654 | 3.414 | 1.2 | 1.6 |

Table 2: Projectile-Metallic plate Materials properties [5]

Once they are defined, several sections need to be created for each one of the materials as "Solid Homogeneous" inside de module "Create Section". These sections have to be assigned to their corresponding parts inside the subgroup "Section Assignment" of the Part module again.

- **Assembly:**
Then, the assembly of the different parts is ready to be performed in order to join them for the simulation. The subgroup "Instances" allows to select them. "Mesh on part" (independent) is selected in order to mesh by parts afterwards.

The objective flat plate is divided in two part which are going to be meshed with different element size in order to save some computational time. For that, we must go to the upper bar to "Partition" inside Tools menu. Then, a vertical line sketch from the type "Face" is made from the point (0.5,0) to (0.5,0.8).

It is highly reccomended to define each part with a different Set a priori in order to make the meshing easier. The following picture shows the assembly of the geometry parts in our practical case:
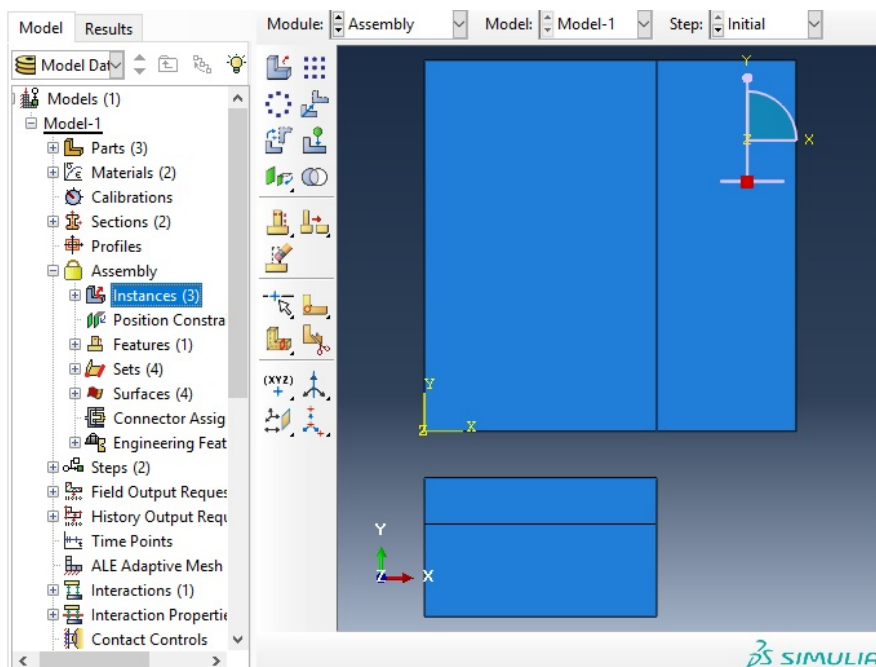


Figure 5: 2D Projectile-Metallic plate assembly

- **Step:**
  Appart from the Initial step defined by default in Abaqus, a new simulation step has to be created. Due to the dynamic problem that we are treating with, it is going to be specified as "Dynamic, Explicit" in Procedure Type inside the "Create Step" module. Here, the simulation time is set, what requires to be consistent with the units by checking constantly the equivalence units in *Table 1*.

  This parameter is very important for a good reproduction of the shock wave travel along the geometry as it is the total time at which our geometry is going to be submitted to the loads or fields assigned to the step. It is different from the total simulation time, which is the one that the CPU lasts to finish the job.

  A time between 5.5 and 6 $\mu s$ must be enough to display the arrival of the shock wave to the back face. Since we can see, the Non Linear Geometric "Nlgeom" option is turned on by default for this kind of step, which means that the solver is able to cover huge rotations of the geometry.

- **Field Output Request:**
  Another interval has to be defined for our simulation independent from the Step time. It is the frequency at which we want to report data from the simulation in terms of spaced time intervals (150 intervals in our case),i.e the number of data outputs of each variable that we are going to extract. The different output variables (Stresses, Strains, Displacement/Velocity/Aceleration, Forces/Reactions...) have to be selected inside the "Edit Field Output Request" module too.

- **Constraints and Interactions:**
  Since the geometry is defined by parts separately, some of them have to be glued in order not to have problems during the simulation. For that, we must define a constraint selecting "Tie" inside the "Create Constraint" module and the master and slave surfaces for the sides which are connected. The rest of parameters are kept to the ones by default.

  Then, we must define the interactions properties in order to avoid the crossing of the impactor through the plate once it arrives. Inside the "Edit Contact Property" module, both Tangential and Normal behaviours from Mechanical menu are set. In our case, we are going to define a Friction Penalty of 0.2 and the "Constraint enforcement method" by default with "Hard" Contact in "Pressure-Overclosure" (make sure the label "Allow separation after contact" is deselected in order to avoid a rebound once the sides crash), respectively. Finally, the iteractions are assigned by surface-to-surface contact type (explicit) and "Kinematic contact method" in the "Edit Interaction" module selecting the sides which are going to be in touch and the previous interaction property defined.

- **Predefined Fields:**
  In this practical case, any loads are acting on the parts because it is the proper interaction between the impactor and the plate what induces the shock wave effects. For this reason, we only have to define a "Velocity" to the Initial Step inside the mechanical category in the "Edit Predefined Field" module. It is going to be translational only in the Y-axis in order that the impact is produced straightly.

  After selecting the whole projectile, we set the an uniform distribution and the value of 0.028 $cm/\mu s$ in the gap of "V2". The arrows which are displayed once preshed in "OK" are indicators of the correct sense of the velocity application.

- **Boundary conditions:**
  They are also defined in the Initial Step as "Mechanical Displacement/Rotation" in category inside the "Create Boundary Condition" module. Then, the degrees of freedom corresponding to the displacement in X-axis (U1) and Rotation (UR3) are constrainted selecting the sides along the central axis of each part.

  In order to avoid a high deformation of the limit region where no load is applied any more, we are going to define an extra boundary condition to fix all degrees of fredom in the down contact surface of the right part of the plate. The following picture shows the final layout of the geometry with the applied fields and BC:
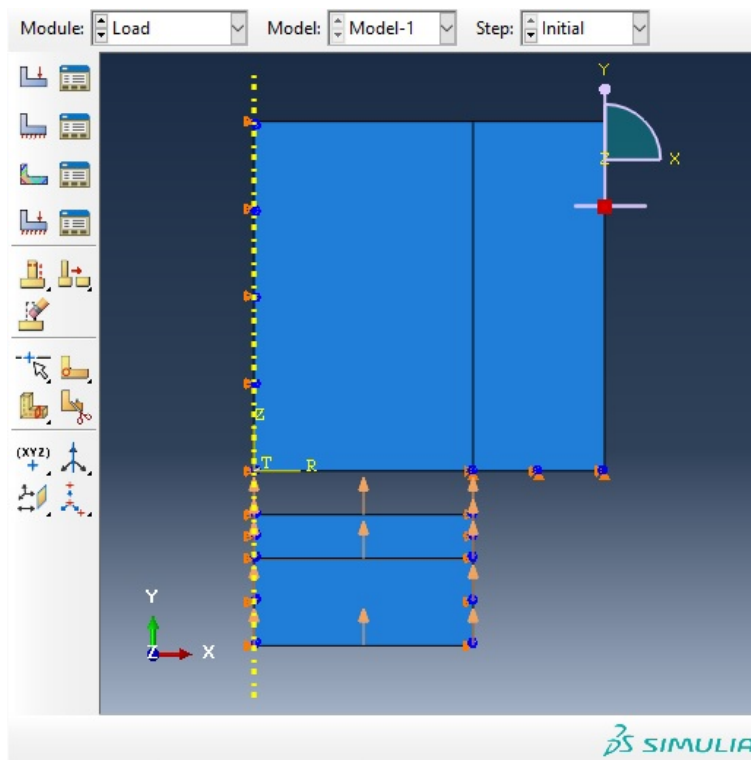
Figure 6: Projectile-Metallic plate fields and constraints

- **Mesh:**

  The last part before simulating corresponds to the Mesh module which has an important role in the quality of the results. Since we have mentioned before, the parts are divided by a partition for computational cost savings. The element type is from "Axisymmetric Stress" Family inside the "Explicit" Element Library (remind to change the library in accordance with the type of simulation). Geometric order is "Linear" and the shape of the elements is quadrilateral "Quad".

  Then, the Element type assigned to our geometry is CAX4R with the element control kept to the ones by default. It is a 4-node bilinear axysimmetric quadrilateral with reduced integration and hourglass control as we can see from the information displayed in the Element type module.

  The next step is to Seed the part Edges, i.e specifying the number of elements in each region. Since we have defined a set for every divided region, we can set the "Local Seeds" setting the approximate element size. This first way of meshing is completely automatic and the aim is improving the quality as soon as we manage to get results (see Optimization of the case). For instance, let us double the size of the elements (0.02 cm) horizontally (the vertical seed is kept to 0.01 cm) for the right region of the plate which is going to be in touch with nothing, leaving the rest to a size of 0.01 cm with the projectile included. Once meshing settings are selected, we proceed to mesh the region individually. The following figure shows a schema of the resultant mesh:
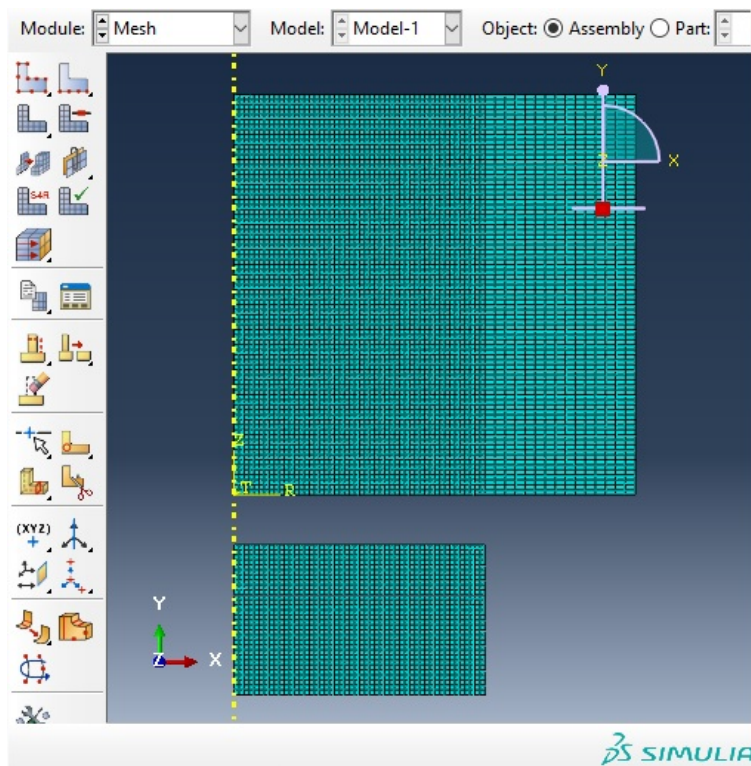
Figure 7: Projectile-Metallic plate mesh

- **Job and Simulation:**
  Once the pre-processing is finished, let us start with the Abaqus simulation. Firstly, a "Job" to submit the case is needed to be created with the only modification of the name. While the iteractions are being performed depending on the frequency previously defined, we are able to open the monitor where errors, warnings and the Data file are shown among other things. The CPU total time needs special attention in order to be computationally efficient, i.e setting an user defined time increment according in the case of an exagerated duration of the simulation.

  If the simulation achieves to finish without any important remarks from the monitor, we will access to the results module or "Output Databases" automatically created during the simulation. Here, we have the possiblity to treat with different kinds of output data and report them into text files in order to perform a better post-processing with a certain programming software.

  Some of the main drawbacks of Abaqus remain in the post-processing part because it is limited when we try to make specific plots, such as 3D representations, Space-Time diagram... Only time histories of a certain variable or a contours plot animation are useful for our case from the software. Then, the option is trying to extract the data by a text file which includes the same number of values of the selected magnitudes that we have set in the Field Output request.

  From the left menu, XY data allows to select a ODB field output from the nodes, elements or integration points (depending on the magnitude) selected from the geometry and represent it against time (space time intervals in which the step time in divided). Then, we can go to "Report" from the upper options bar to select the previous XY data created. For the Setup, we must save the file with the .txt extension with the option Separate table for each XY data inside "Output format" enabled. Here, it is possible to specify the number of significant digits, number format and some other options for the text file appearance (very important to simplify the reading with the programming software).

  In our case, Matlab is used to treat with the output data. A code is done to read the entire text files and to store the variables in a set of matrices which are going to be used for plotting. This tool provides us a subtancial help in analysing and understanding a shock wave simualation by the possibility of building a variety of graphs. The Matlab scripts are included in the section Annex at the end of this report with some explanations of the code.

### 3.2.2   Post-processing

This section has the goal of validating the previous model through several output data which can be compared with the analytical theory of laser shocks. The main data extracted from the resultant file are:

- **Contour plot Animation:**
  It displays the evolution of the shock wave along the objective plate once the impact is produced and the corresponding reflection in the back face, by the previously defined time increments until completing the whole dynamic step. Different variables can be set in the upper bar options (multiple choice menu) so as to plot their contours on the deformed-meshed shape of the geometry (Plot...Contours...On Deformed Shape).

  Concerning our aim, we have to check that the hydrodynamic pressure is transferred by the expected way through the different interfaces that the shock wave can encounter. According to the definition of the stress and making the assumptions explained before, we can treat the main stress that our geometry undergoes ($\sigma yy$) as the opposite of that term of pressure. Below, we have shown a list of captures showing the evolution of this variable with respect to time:
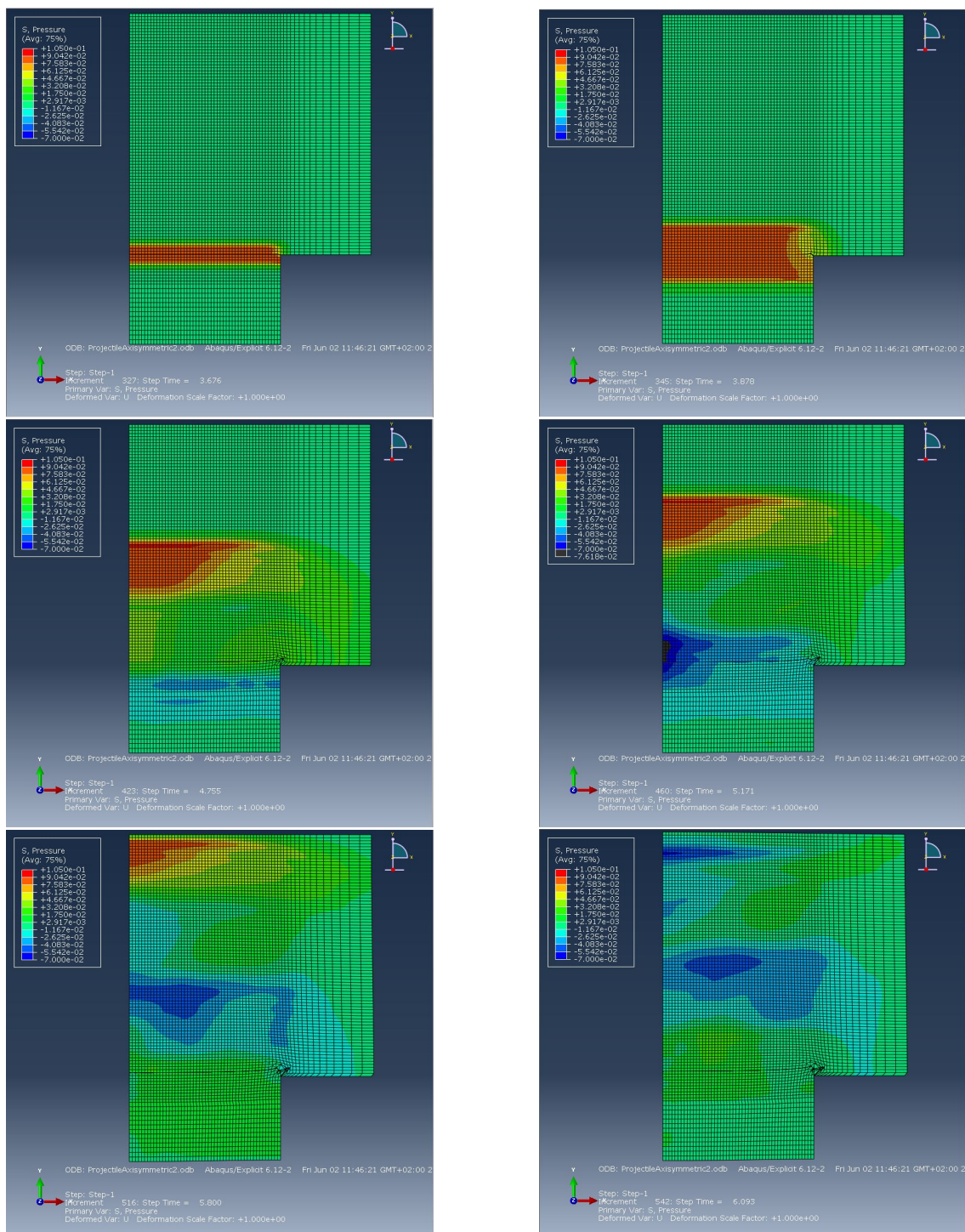


Figure 8: Evolution of pressure after the projectile impact

The range of contour is fixed between -0.07 and 0.105 Mbar so that the wave propagation is visible. The impact is produced when the projectile gets the contact zone with the plate, which has been modelized in order to avoid the crossing (1st row, 1st picture). The wave is propagated towards both senses what validates our assumption of conservation (1st row, 2nd picture). Then, we can obserbe the transition of the shock wave along the axisymmetric axis of the plate (2nd row, 1st picture) until the arrival to the end of the geometry (3rd row, 1st picture). As soon as it passes an interface either between materials junction Al-Ta in the projectile (2nd row, 2nd picture) or the reflection in the back face, the tensile stresses are produced and represented by the blue regions (3rd row, 3rd picture).

Here, it is also possible to note the origin of the called 2D effects. They represent the reflection of the shock wave once it arrives to the extremes sides during its propagation. The apperance of this phenomena is observed in a non-constant/uniform shape of the wave (follow the blue contours in the animation) travelling along the material and they change the behaviour of the wave propagation. In addition, it also induces shear stresses which produce changes in the pressure and velocity evolutions with time comparing with the analytical case in the following section.

- **Time records:**
  Conservation laws, defined in the theory background section, have to be accomplished in our model. That means continuity needs to be respected once the shock wave passes through an interface. For this purpose Time histories are performed recording the evolution in time of different variables choosing one element (Von Mises stress, Stress components, Hydrodunamic Pressure...), or node (Displacement, Velocity, Acceleration...). Several time recordings are made in the following nodes:
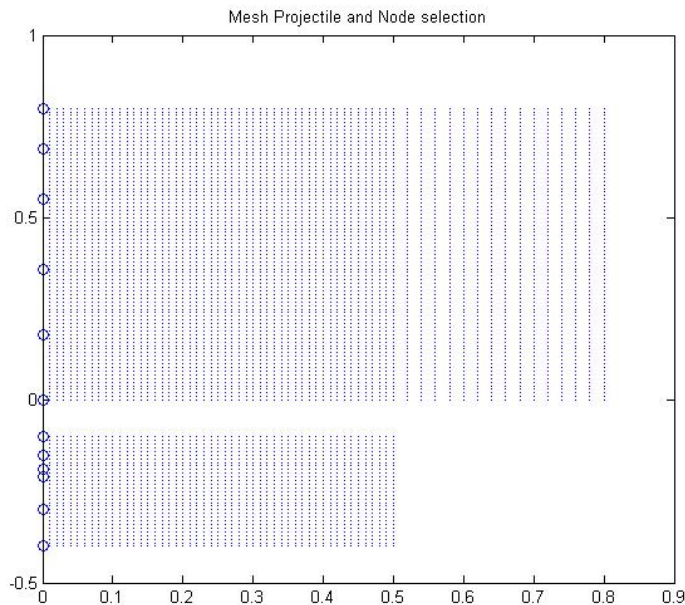


Figure 9: Selected nodes for output data

First of all, we are going to prove the laws with the evolution of velocity in the nodes close to the interface region, i.e where two parts of the geometry make contact to each other. The same tendency is expected to see after the impact (at $\simeq 3.6\mu s$) between the plate and the tantalum part of the projectile because the wave is propagated by the same way in both parts:
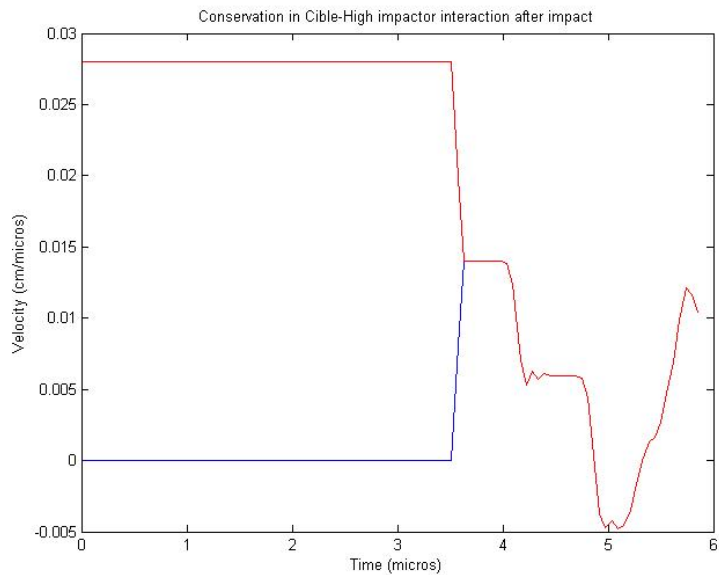
Figure 10: Continuity in the interface upper projectile-plate (Ta-Ta)

A more difficult task is observing the continuity between two materials interfaces because they have different properties, such as density or sound velocity of propagation. The wave is not going to travel the same distance in both of them, so it seems to be necessary enlarging the size of the elements for the "fastest" material (the one with higher sound velocity) in order to have a node of this material at the same time instant to prove the conservation.
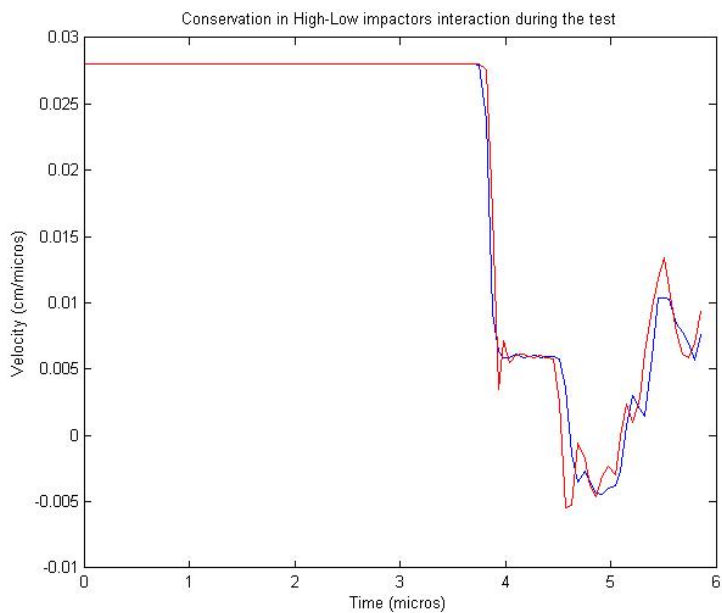


Figure 11: Continuity in the interface lower-upper projectiles (Al-Ta)

Since it is mentioned above, another important variable to record its evolution with time is the pressure, which is represented in a 3D plot (Space vs Time vs Pressure) taking the list of corresponding elements to the previous nodes captures. This plot allows us to detect the reflections of the wave with the interfaces of materials or with the plate back face inducing tensile stresses (considering them as negative values of pressure), which correspond to the red-circle points:

Time histories also allow to validate the model by comparing both pressure and velocity evolution versus time with the theoretical model, whose solution coming from the application of the shock polar curve is taken from bibliography. In the Validation section, we are going to discuss how accurate our model has been, explaining more in detail the differences that we find with the theoretical model.
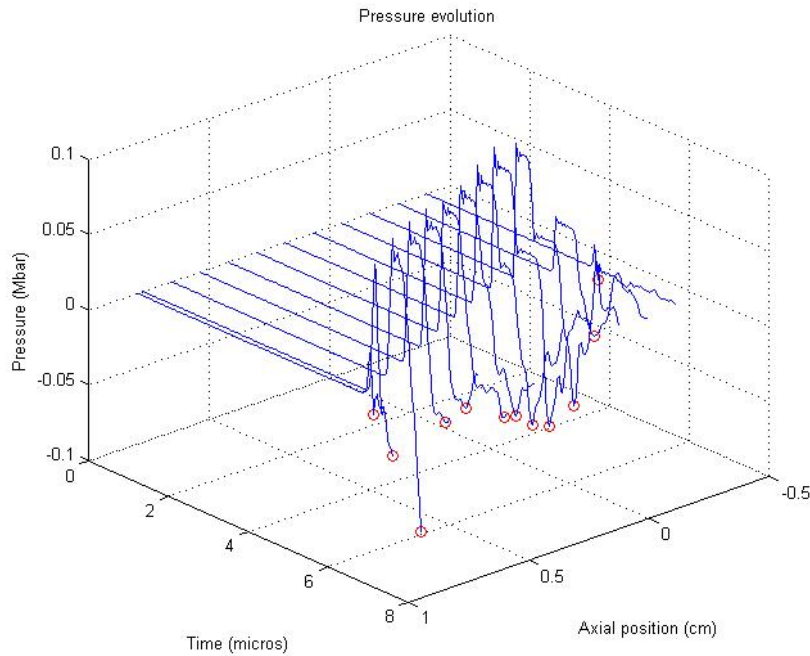
Figure 12: Pressure evolution versus time for different elements of the geometry

- **X-T diagram:**
  In order to magnify the contrast of pressure when a reflection is produced, the X-T diagram is a 2D representation of the axial position of elements in the whole time period plotting contours of pressure which are ranged by a color bar.

  It is an evidence that the quality of resolution is higly dependent on the number of elements selected along the Y-axis (direction of the projectile movement). Here, we can see the diagram for the set of elements previously selected as well as selecting the whole line of elements attached to the cental axis:



(a) A few elements

(b) Whole line of elements
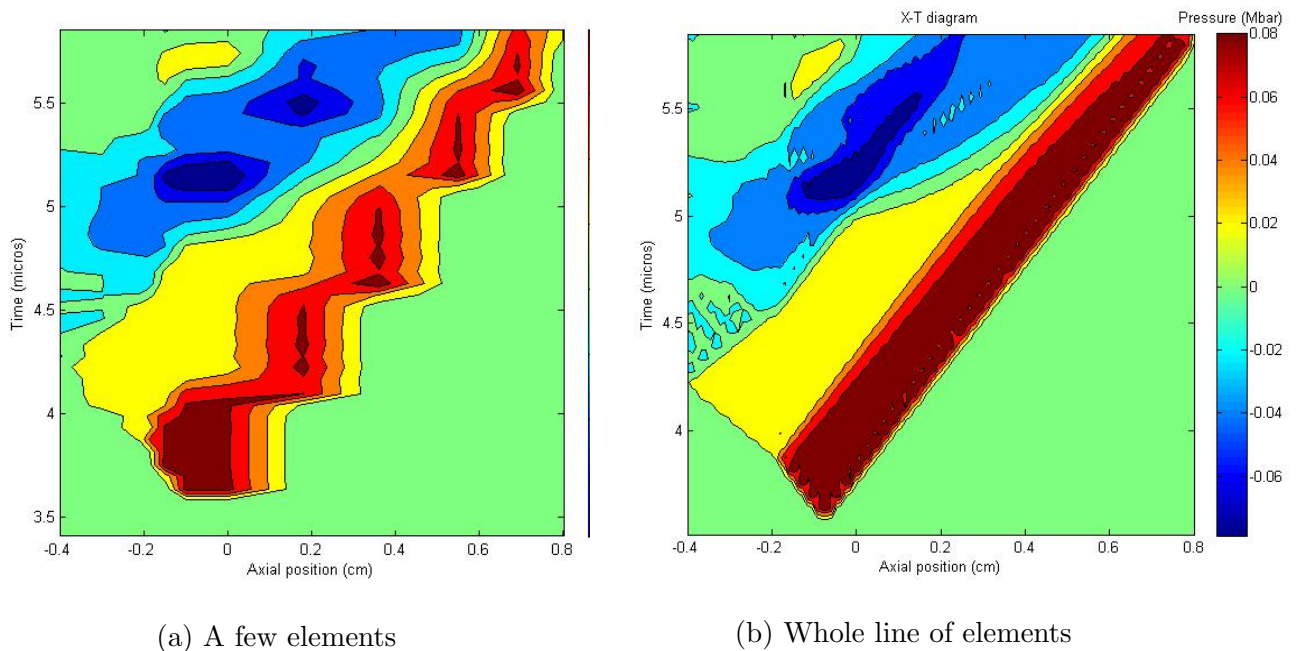
Figure 13: X-T diagram with pressure contours

It is possible to detect the travel of the shock wave from the front to back face once the impact is produced. However, a longer step time duration $(> 5.8\mu s)$ is needed to reproduce the back of the wave and the corresponding tensile stress. It is important to be careful when increasing the step time because a long exposure of the plate to the impact of the projectile can induce a high deformation of the elements and wrong pressure/stress values appear.

There are two possiblities: one is editing the step time and start the simulation again or the other one is concerning to create another step with the increase in the simulation time and restart the simulation from the point it had ended. This second option allows to analyze the reflection more in detail increasing the frequency of data reporting in the "Field Output Request" module.
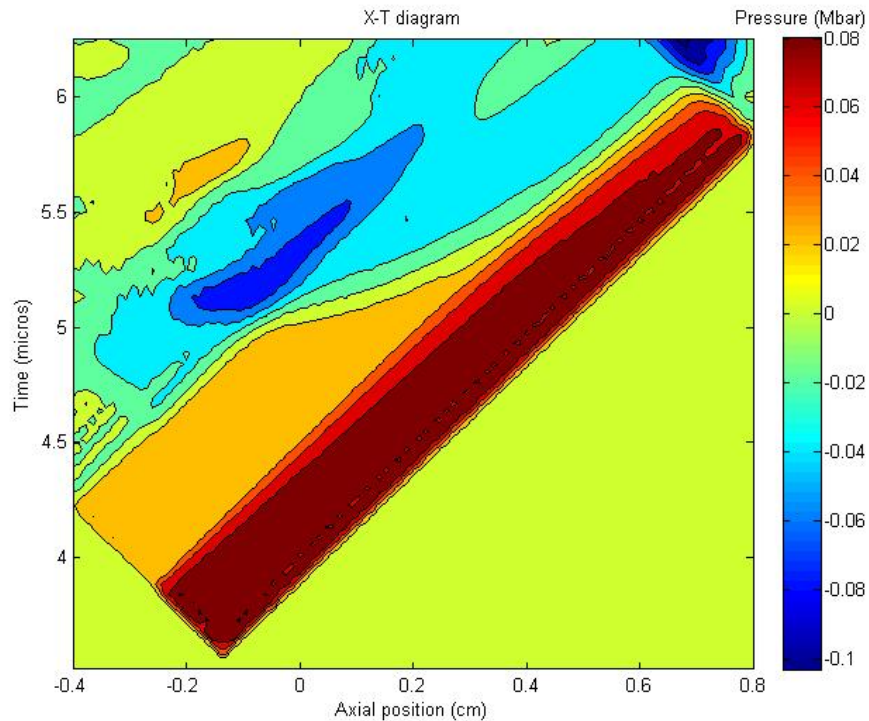


Figure 14: X-T diagram with increased step time
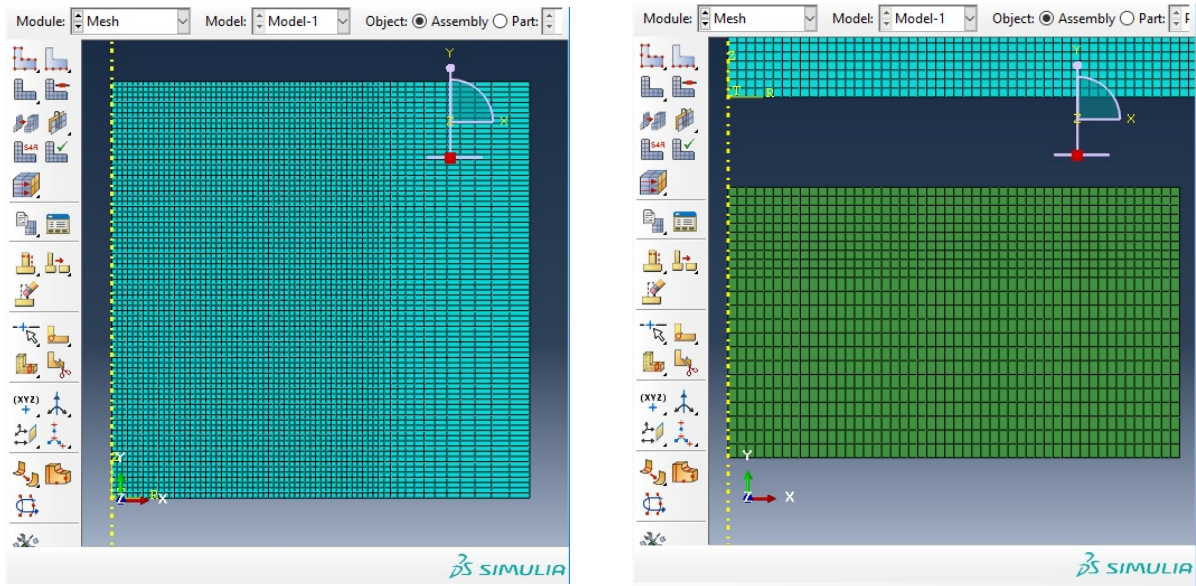
### 3.2.3   Optimization of the case

Since we have seen, several techniques can be performed in order to get a better resolution of the output. For example, increasing the time interval of the step to 6.25 $\mu s$ allows to observe the reflection of the wave in the back face. Consequently, we have increased the frequency of reporting data in the "Field Output Request" to have more time instants where the results are taken. These settings are only concerning the visualization of the results and do not change them.

Let us introduce the other changes in the pre-processing in order to get a more accurated results to the theoretical ones, what help us to validate the model. The aim of these improvements is also trying to correct the discontinuities that can appear due to the huge deformation of the elements when the impact takes place and which are visible in the animation of contours.

We are going to avoid defining a constraint to glue the parts in contact by merging the nodes in the assembly module. The command can be found inside "Instance" in the upper bar, whose name is Merge/Cut Instance. This creates a new joined part from the selection of the instance to be merged. The programme automatically detects which are the nodes in touch if we choose the options "Merge" and "Mesh" in Operations with a certain tolerance (we keep the one by default). Then, the rest of the pre-processing settings need to be redefine to the new part, such as the interactions, boundary conditions or predefined-fields.

However, the previous setting requires that the final mesh has to be defined in advance. Here, we can introduce new changes for the element size in the region which would not be in contact with the projectile. Now, we define a geometric rule in order to increase horizontally and towards the right the element size parting from 0.01 cm until 0.04 cm in the extreme. In the module "Seed Edges" and taking the side by side, a single bias inside "Local Seeds" has to be set specifying the maximum and minimum sizes and changing the sense of growth in Flip bias if it is neccesary. In addition, a change in the size of the Aluminum elements is also performed in order enlarge them vertically and be able to measure the magnitudes in the nodes at the same time instant:

(a) Plate

(b) Projectile

Figure 15: Optimization of the mesh

The discontinuity is still appearing, but our model is good enough to make it valid as it does not affect too much to the wave propagation. Another techniques are left as proposals to try. For instance, defining a limit of maximum strain $\epsilon_{max}$ so that the programme delete such elements of the discontinuity which are suffering a high deformation, or trying to set a time increment (different from the intervals of data reporting) in the step in order to make the simulation slowly because the programme cannot approach a high-velocity impact performed directly with an excessive deformation rate of the elements.

### 3.2.4 Validation of the case

Every jump in pressure/velocity correspond to the pass from one state to another. The states are regions enclosed by the reflections of the wave through the interfaces or the faces and are built with the X-T theoretical diagram. The time observed in the analytical X-T diagram has to be increased the time it lasts until the impact happening, i.e around 3.6 $\mu s$.
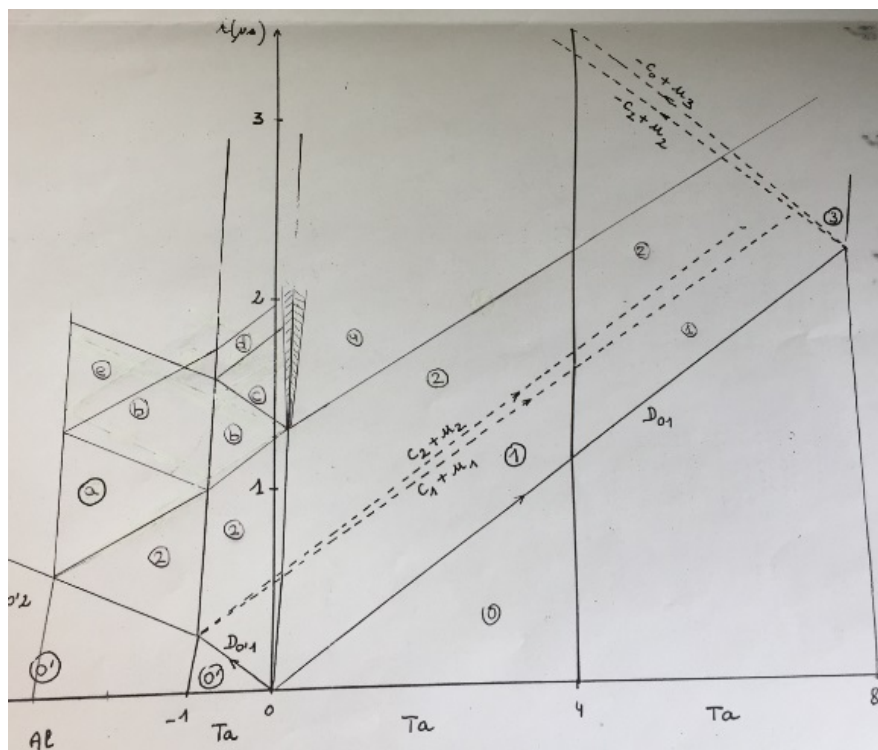


Figure 16: Analytical Space vs Time diagram for the projectile impact case

Then, the data is transferred to the polar curve in order to see the conditions of pressure and velocity that every state has and to compare with the jumps in the time history.
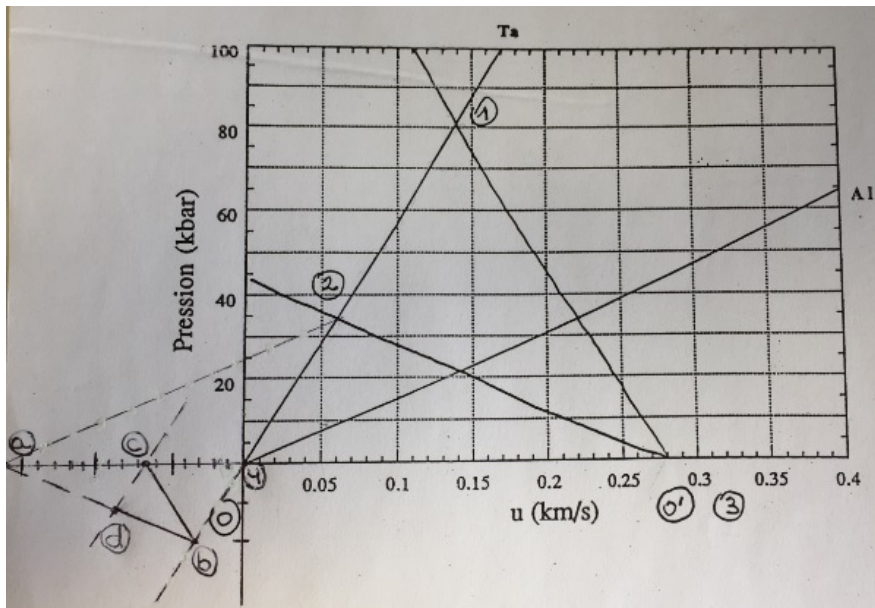
Figure 17: Analytical Pressure vs Velocity diagram for the projectile impact case

Two examples of the time histories are recorded in the main elements and nodes of our geometry as it is specified in the legend. These are the lower ("Extreme Projectile") element in the impactor, the elements at both sides of the interface Al-Ta ("Interface Al", "Interface Ta"), the Ta ones from the plate and the impactor which are in touch after the impact ("Interface projectile", "Interface Cible") and the last ones are in the middle ("Middle Cible") and at the extreme ("Extreme Cible") of the objective plate:
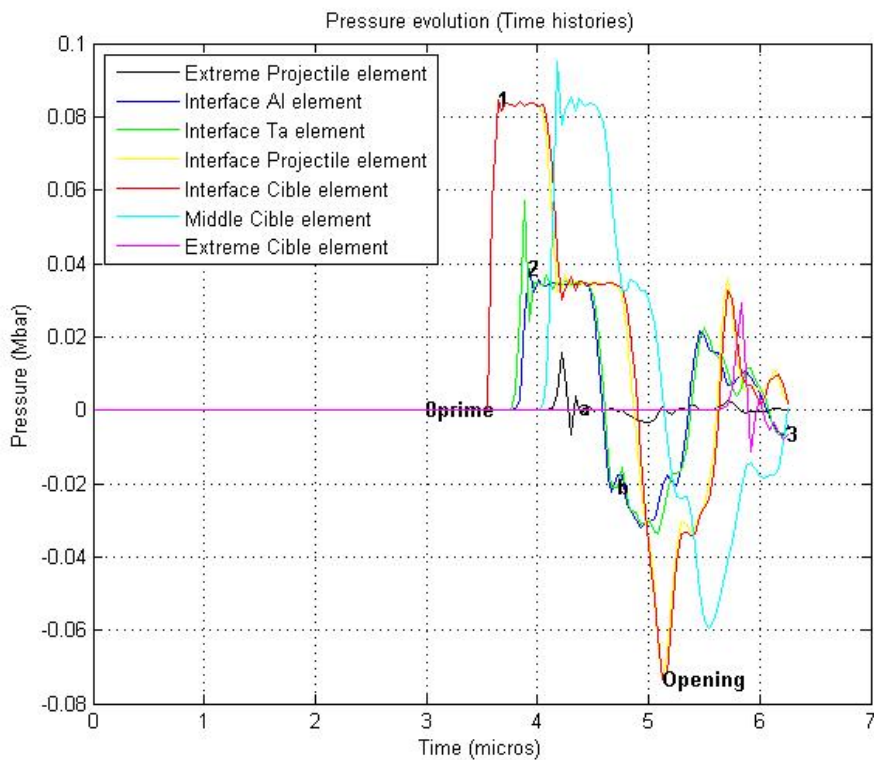


Figure 18: Pressure vs Time for the main elements of the geometry
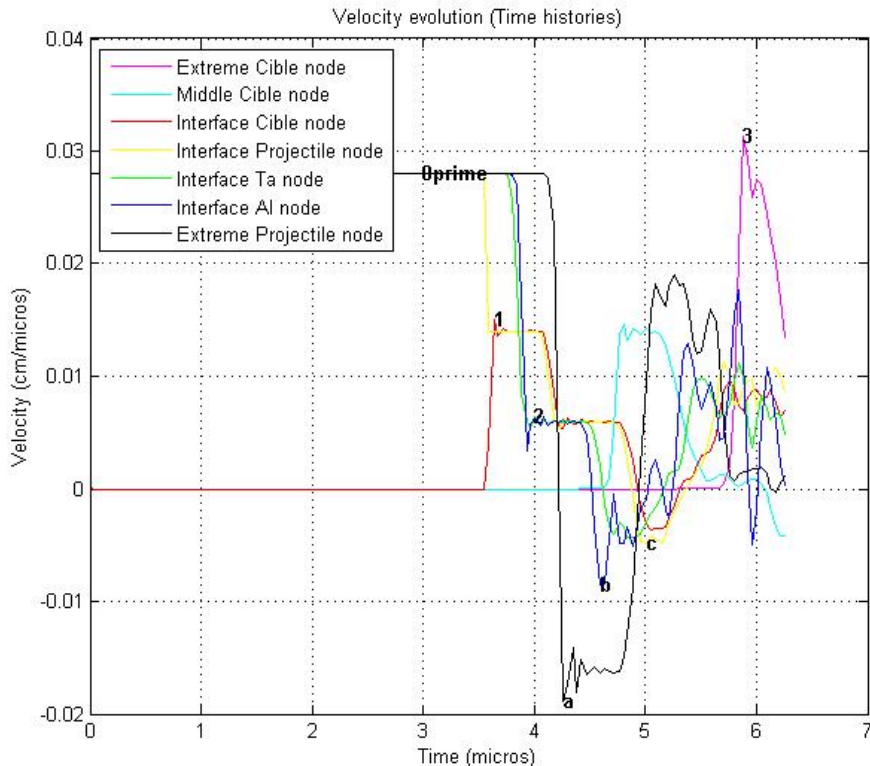
Figure 19: Velocity vs Time for the main elements of the geometry

Let us start with the elements of the projectile. In the extreme, the black element goes from state 0' which is at zero bar and 0.028 cm/$\mu s$, then a brief crossing inside the state number 2 (at 0.034 Mbar and 0.006 cm/$\mu s$) explains that peak which tends to grow, but it passes quickly to the state "a" at zero presure again. Initially, this element has the imposed velocity in the pre-defined field of the pre-processing. Then, the negative value in the state "a" corresponds to a change in the direction of wave propagation due to the reflection with the back face of the projectile. From a certain time instant, 2D effects appear, so the behaviour of the wave propagation changes and this way of validation is not consistent enough.

The two elements in the projectile interface, the blue of Al and the green of Ta, are joined during the evolution of time histories. While the green distribution has a brief tendency to grow towards state 1 (at 0.08 Mbar and 0.014 cm/$\mu s$), the blue one passes directly to the state 2. Then both cross to the state "b" which corresponds to the tensile stresses (negative pressure) appearing after the reflection with the back face in the way of a pressure release which allow to identify where the damage is produced. The projection of the wave release towards the Ta plate will produce the separation/opening of the interface Ta-Ta modifying the velocity of the material as a new state of zero pressure (air) is created in the middle of both separated sides. Concerning the velocities, states 0' and 2 are perfectly recognised as well as the negative value of state "b". Again, 2D effects appear inducing changes in the time histories of pressure and velocity.

For the interface between the projectile and the plate (Ta-Ta) the previous separation is produced. It affects to the elements taken at both sides, yellow and red distributions, one from each part. They follow similar tendencies passing through state 1 and 2 until arriving to the negative pressure zone and the opening of the interface. However, this phenomena is difficult to be represented in the modelling of the case, so that only a huge drop in pressure is produced. Both elements should tend to zero pressure as it can be checked for states "c" and 4 at the left and right sides of the opening, respectively. For velocities, the passing trough the negative values of state "b" followed by "c" is observed. It is important to remark that the states labeled by letters are all given in the projectile.

Finally, the elements of the plate, cian in the middle and magenta in the extreme, follow different tendencies. Both parting from state 0, the first one passes through state 1 and 2 theoretically, but the appearance of 2D effects make impossible to detect this last state. The second one has a brief crossing through state 1 which explains the peak, but it tends to zero pressure for state 3 (at 0 Mbar and 0.028 cm/$\mu s$) again. For velocities, only state 1 for the cian element and state 0 for the magenta one, which correponds to the pre-defined velocity.

The following pictures correspond to the superposition of the pressure and velocity time histories of a few elements/nodes (in the middle of the objective plate and in the interface Al-Ta), obtained from both the analytical and the numerical cases, which validate our model definitively:
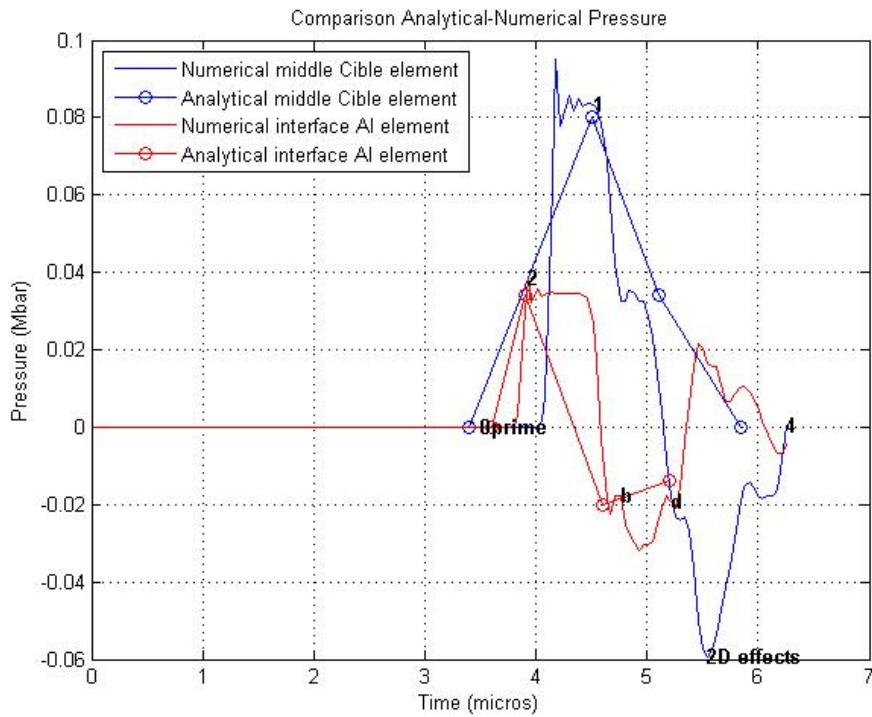


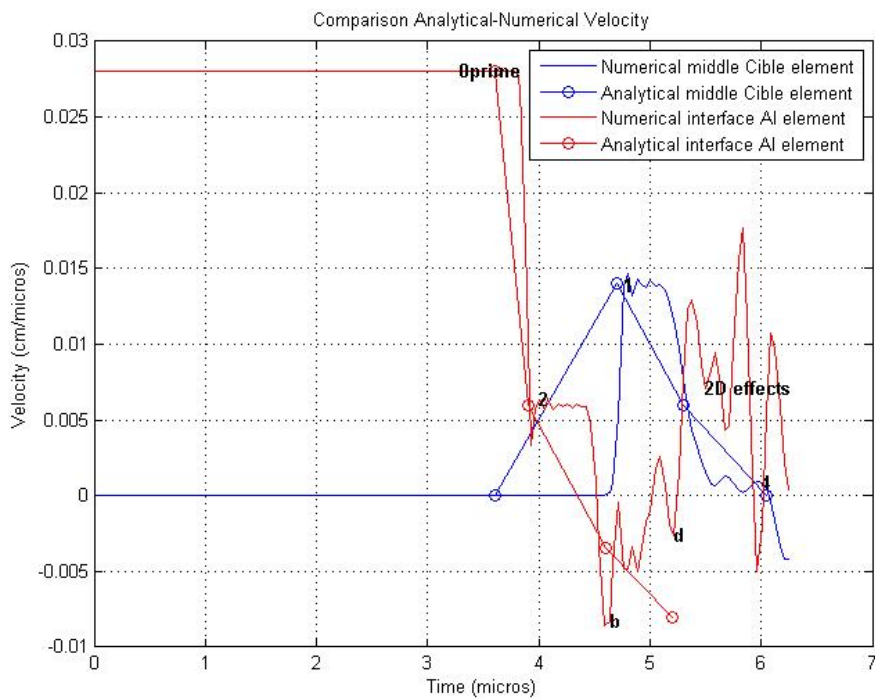Figure 20: Analytical-numerical pressures for the impact test



Figure 21: Analytical-numerical velocities for the impact test

We can conclude that our model is validated for the time history between the impact and the moment when the 2D effects appear. The study of these effects corcerns to a more detailed analysis of the laser shock phenomena. For that, we only consider 1 direction of the wave propagation.

## 3.3 Pressure distribution over a flat plate with coating

In this case, the problem consists on the application of a direct pressure distribution over one side of an aluminum flat plate during a certain time. The action of the pressure effort induces a shock wave which travels along the geometry in the direction of the symmetric axis until arriving to the region of the coating made of a certain fiber, approximation by epoxy. The aim is testing the adherence of both materials by laser shock simulation, what is commonly applied in the aeronautic industry, assuming a purely hydrodynamic behaviour.

In order to build the analytical model for a pressure distribution such as our case, several main points must be taken so that we can reproduce the shape of the pressure profile as good as possible. In this case, it will be necessary to make difference between the compression and release wave (dashed-lines) because many projections are going to be crossed between them. The following picture shows an example of what we need to have in mind:
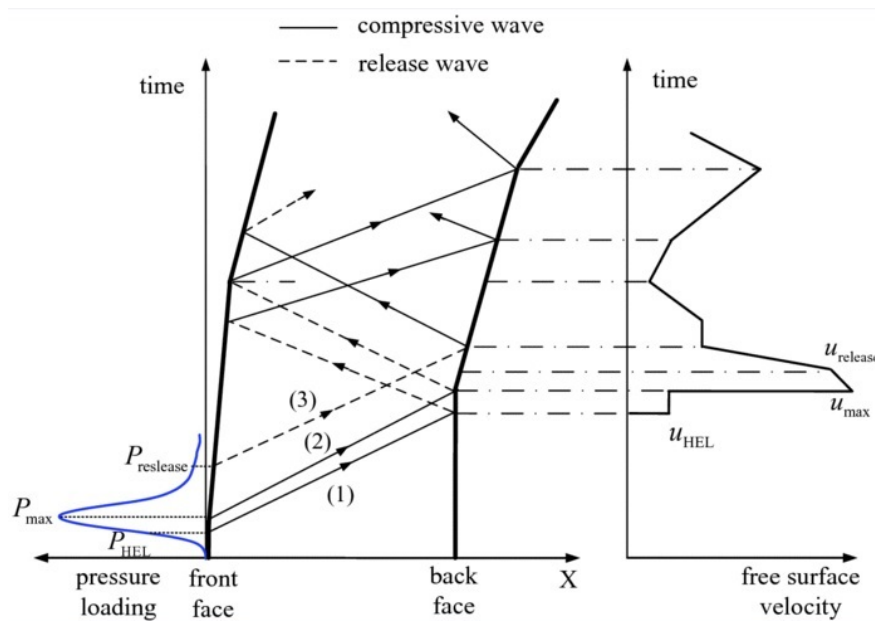


Figure 22: Example of analytical X-T diagram with the application of a pressure distribution

The following steps of the report include the procedure performed for the definition of Finite Element model in Abaqus (geometry, material properties, time step definition, load and boundary conditions, interaction properties, mesh, output fields...).

### 3.3.1 Pre-processing and Abaqus Running

- **Geometry:**
  The approach of the case will be similar to the previous one, i.e considering the geometry axisymmetric which permits to avoid modelling the whole plate. The dimensions of the remaining aluminum plate are 3x1 mm with the addition of a 3x0.5 mm coating. Since it is a flat plate with negligible thickness and due to the sense of the shock wave propagation, the problem can be treated as 1 dimensional:
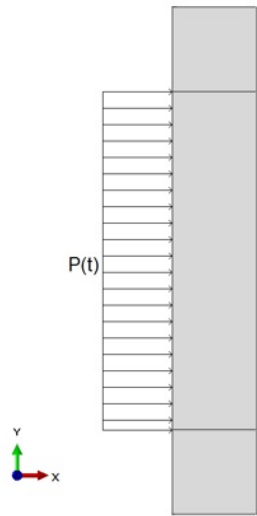
Figure 23: Geometry of the flat plate case [4]

The different parts of the problem are defined in the module "Create Part" selecting Modelling Space "Axisymmetric" (Y-axis by default in Abaqus), Type "Deformable" and Shape "Shell". Due to the predefined settings of the axisymmetric mode, the rectangle is built lying along the x-axis.

The resultant flat plate is wanted to be divided in two parts to be meshed with different element size in order to make difference between the loaded and the free loaded regions. First, we should perform a partition from the Tools bar in the horizontal upper and lower sides selecting "Edge" in type and "Enter parameter" in method. The partition is made both for the plate and the coating in the limit of the region where the load is applied. A number between 0 and 1 is required, so the limit point should be at 2/3 from the origin located at the central axis. Then, a vertical line between the previous created points is made in a sketch ("Face" type) from "Partition" inside Tools menu again.

- **Materials:**
The two materials which are put in place in this case are Aluminum and Epoxy. We are going to compute several simulations modifying the material assignments for every part in order to see the difference between different cases: Al-Al and Al-Epoxy, both with the properties of its corresponding equation of state in the Property module.

Another properties to be defined include the density, inside the General menu, or the isotropic elastic properties, such as Young modulus or Poisson ratio, inside the Mechanical menu.

The following table contains the values of every variable which must be transformed into the right units:

| Material | $\rho_0(kg/cm^3)$ | $E(MPa)$ | $\nu(-)$ | $c_0(m/s)$ | $s(-)$ | $\gamma_0$ |
|----------|-------------------|----------|----------|------------|--------|------------|
| Aluminum (Al) | 0.0027 | $70e3$ | 0.33 | 5328 | 1.339 | 2.09 |
| Epoxy | 0.001186 | 1600 | 0.4 | 2730 | 1.493 | 1.13 |

Table 3: Flat plate materials properties [5]

Once they are defined, several sections need to be created for each one of the materials as "Solid Homogeneous" inside de module "Create Section". These sections have to be assigned to their corresponding parts inside the subgroup "Section Assignment" of the Part module again.

- **Assembly:**
By pressing in "Instances", the assembly of the different parts is performed in order to join them for the simulation. The "Intance type" must be selected depending on if we want to mesh on part (dependent) or mesh on instance (independent). The following picture shows the assembly of the parts in this practical case:
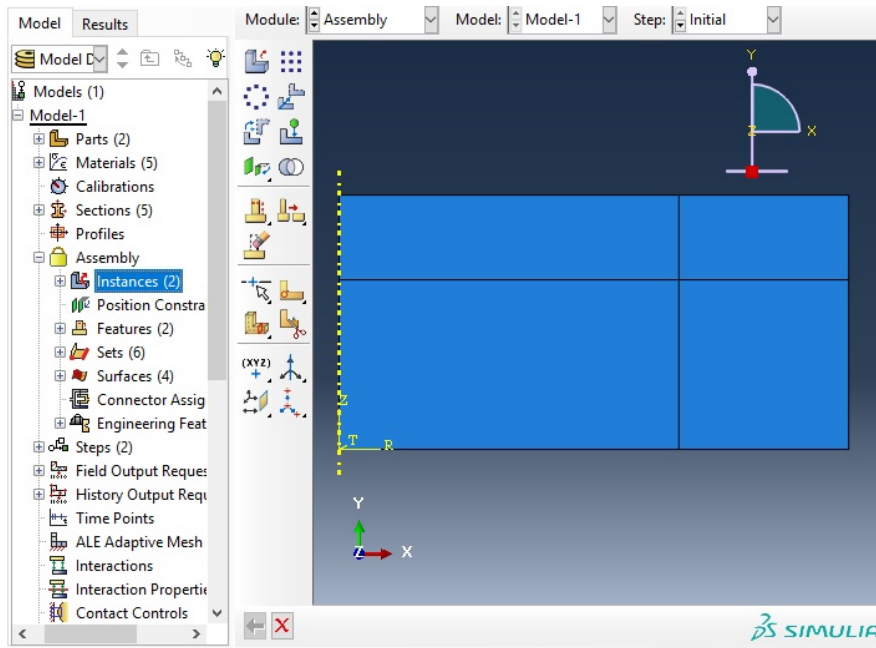
Figure 24: Half flat plate assembly with coating

- **Step:**
  Appart from the Initial step defined by default in Abaqus, a new simulation step has to be created. One can think that the case is static because the application of the load is made directly without movements of the parts. However, the effects induced is what makes necessary to set the problem as "Dynamic, Explicit" in Procedure Type inside the "Create Step" module.

  Again it is required to be consistent with the units by checking constantly the equivalence units in *Table 1*. For this case, a time between 0.0004 and 0.0005 $ms$ with the Non linear geometric option enable again.

- **Field Output Request:**
  The frequency of reporting data from the simulation is set to 200 spaced time intervals for this case. The different output variables (Stresses, Strains, Displacement/Velocity/Aceleration, Forces/Reactions...) have to be selected inside the "Edit Field Output Request" module too.

- **Constraints:**
  The parts (plate and coating) which are going to be in contact must be glued in some way in order not to have problems during the simulation. For this case, a more efficient way to fix the parts comparing with the "tie constraint" is approached. It consists on pressing the command "Merge/Cut Instances" from the icons to the left of the display inside the Assembly module.

  Basically, it works in the same way as the previous case: it joins the nodes of the parts which are in direct contact during the simulation choosing the options "Merge" and "Mesh" in Operations.
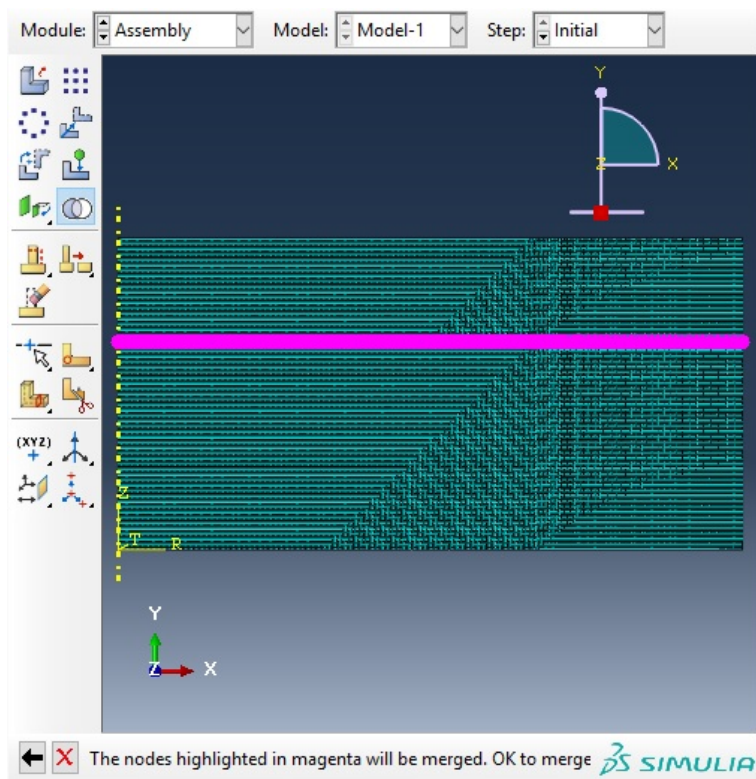
Figure 25: Nodes merging of the parts

- **Mesh:**
Since it was mentioned before, we have only considered the half of the complete geometry and the resultant parts are even divided by a partition to save some computational time. The element type is again CAR4R corresponding to "Axisymmetric Stress" family: 4-node bilinear axysimmetric quadrilateral with reduced integration and hourglass control.

For this case, we opt for setting a geometric rule (single bias) inside the "Local Seeds" module to increase horizontally the size of the elements towards the right. It starts from 0.01 mm till 0.04 mm ("Sizing controls") from the partition to the extreme (check "Flip bias"), for both the plate and the coating. Once meshing settings are selected, we proceed to mesh the region individually. The following figure shows a schema of the resultant mesh:
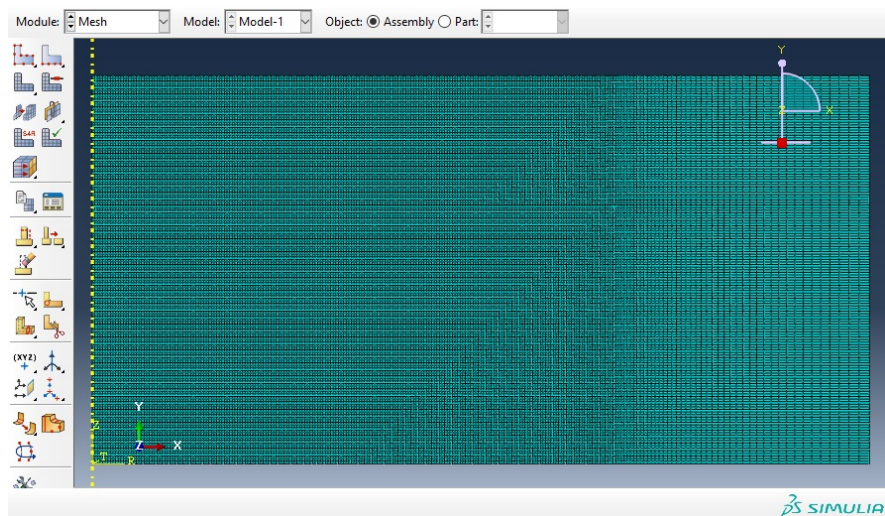


Figure 26: Flat plate mesh

- **Loads:**
Due to the simplification of the geometry, the load is applied from the origin to the limit of the partition for the half of the geometry. Since it consists on a pressure distribution extracted from a datafile (probably from experience), a "Tabular" Amplitude is needed to be set inside the module "Create Amplitude". Here, every point of the pressure curve is defined by writing its normalized value at its specific time in the corresponding units.

The load is created in the defined step, the down side of the plate is picked and the "Magnitude" and "Amplitude" labels are filled with the maximum value of pressure (540 Mpa) and the previous amplitude inside "Edit Load", respectively. The distribution is kept to "Uniform".

Make sure that the total simulation time does not exceed the application time of the load in order not to obtain wrong effects in the results. The following picture shows the normalized pressure distribution which is applied to the geometry:
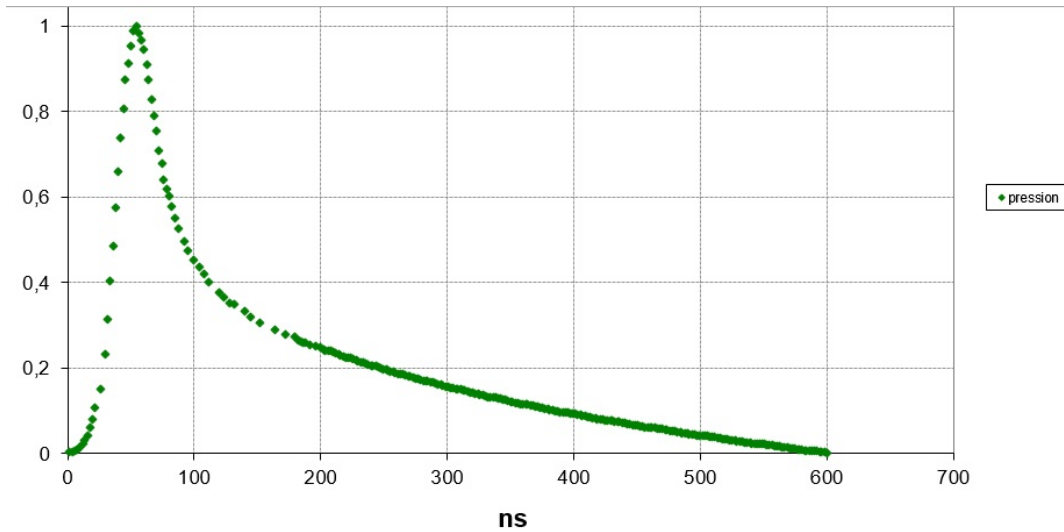


Figure 27: Pressure application

In addition and before the previous case, we are going to simulate every case with a constant amplitude pressure load (maximum value) in order to check the behaviour of the shock wave in our geometry. This is going to be better explained in the Postprocessing section of the case.

- **Boundary conditions:**
  Defined in the Initial Step as "Mechanical Displacement/Rotation" in category inside the "Create Boundary Condition" module, both displacements in X (U1) and Y-axis (U2) as well as the rotation (UR3) are constrained for the extreme sides of both parts.

  Again, we try to reduce the deformation due to the discontinuity in the limit which separates the free loaded region, by defining an encastre in the right part of the down side of the plate. The following picture shows the final layout of the geometry with the applied fields and BC:
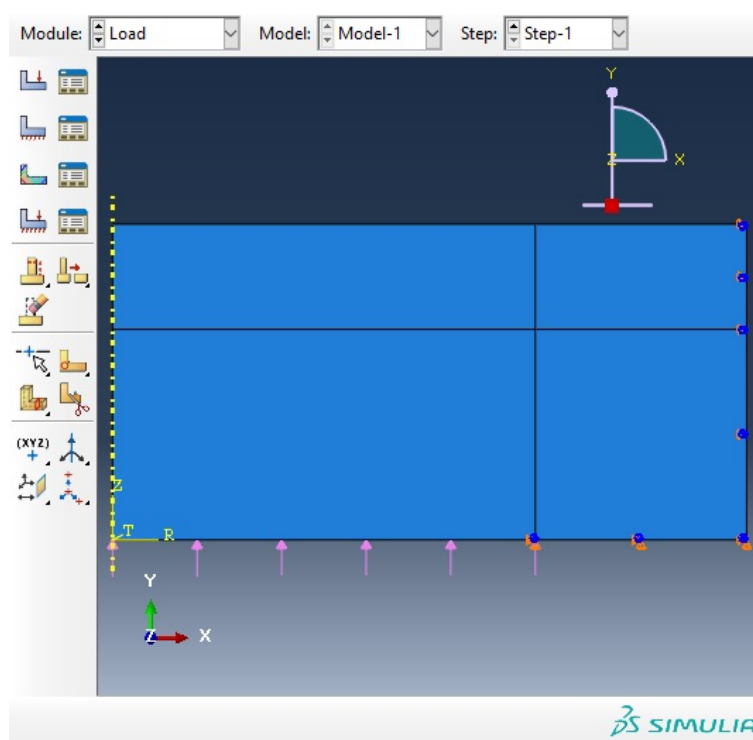


Figure 28: Flat plate loads and constraints

- **Job and Simulation:**
  The case is submitted in several Jobs recorded with different names changing the conditions of materials or load application, which are previously defined, for every case. As soon as the simulation finishes without remarking errors in the monitor, we are able to access to the output database in order to treat with the results. The post-processing procedure is performed by the same way as the previous case, mainly with some Matlab scripts created for the case and also included in the Annex.

### 3.3.2  Post-processing

- **Contour plot Animation:**
  Since it was mentioned before, simulations with uniform amplitude of pressure are performed in order to observe that better and to make a first validation of the case. Below, we can observe several captures showing the evolution of pressure in the Animation:
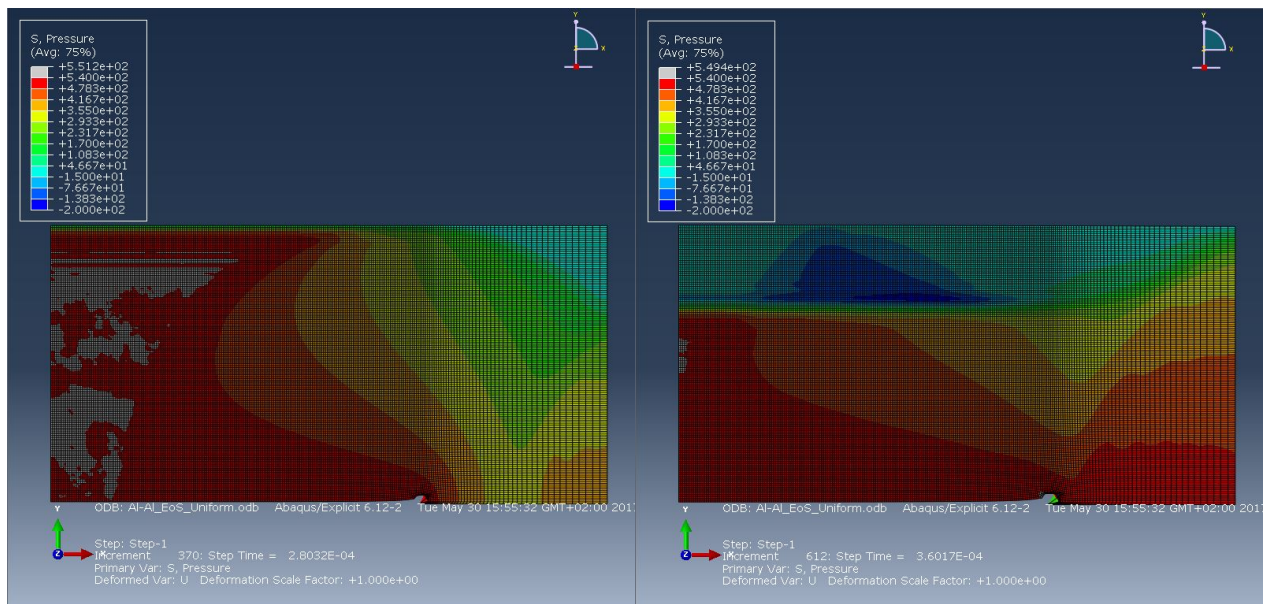


Figure 29: Pressure evolution in interface Al-Al with EoS (Uniform loading)

The range of contour is fixed between -200 and 540 MPa so that the wave propagation is visible. The first picture of each case shows the arrival to the back face of the plate for the Al-Al case and the arrival to the Epoxy coating for the other one. After them, the reflection of the wave is produced inducing tensile stress by different ways.
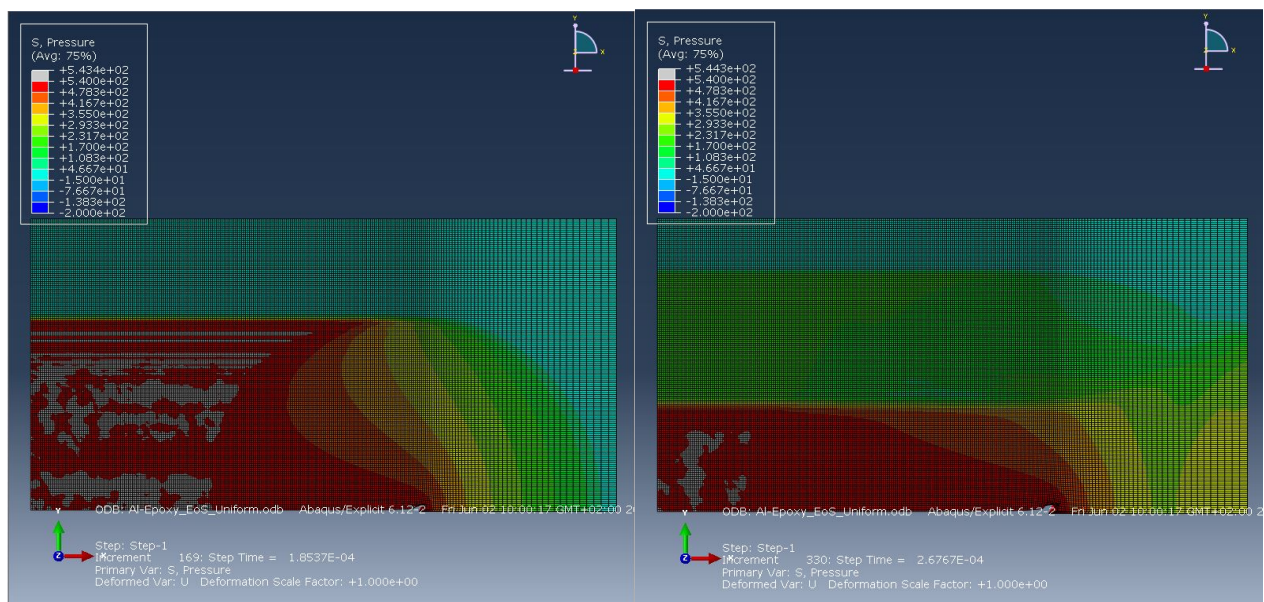


Figure 30: Pressure evolution in interface Al-Epoxy with EoS (Uniform loading)

Both cases have a common denominator: undesirable stresses are produced where the partition is made due to a discontinuity. Some ways to solve this meshing incompatibility can be approached, such as the ones proposed in the optimization section of the previous case.

Except from that, it seems that the wave is well propagated along the plate in the axisymmetric direction with the difference in the instant when tensile stresses appear because of the coating material. Before passing to the next case with the pressure distribution, we must continue validating this simple case with the time histories of the main elements/nodes of the mesh.

- **Time records:**
It is also necessary to check if the elements undergo the corresponding value of pressure set in the "Create Load" module, what verifies the assumption of the hydrodynamic part of pressure. The time histories recordings of the pressure versus time show the pressure profile initially defined, which has to be perfectly represented in the case of Al-Al interface for every element. However, the addition of the epoxy coating change the pressure profile as soon as the shock wave gets the interface.

The way to validate the previous distribution is checking if the values of pressure and velocity coincide for every state comparing with the theoretical case. For that, we must make use of the P vs Time and X-Time diagrams built by hand following the theory. In this case, only a compression wave from zero pressure till the maximum amplitude is mantained along the time.
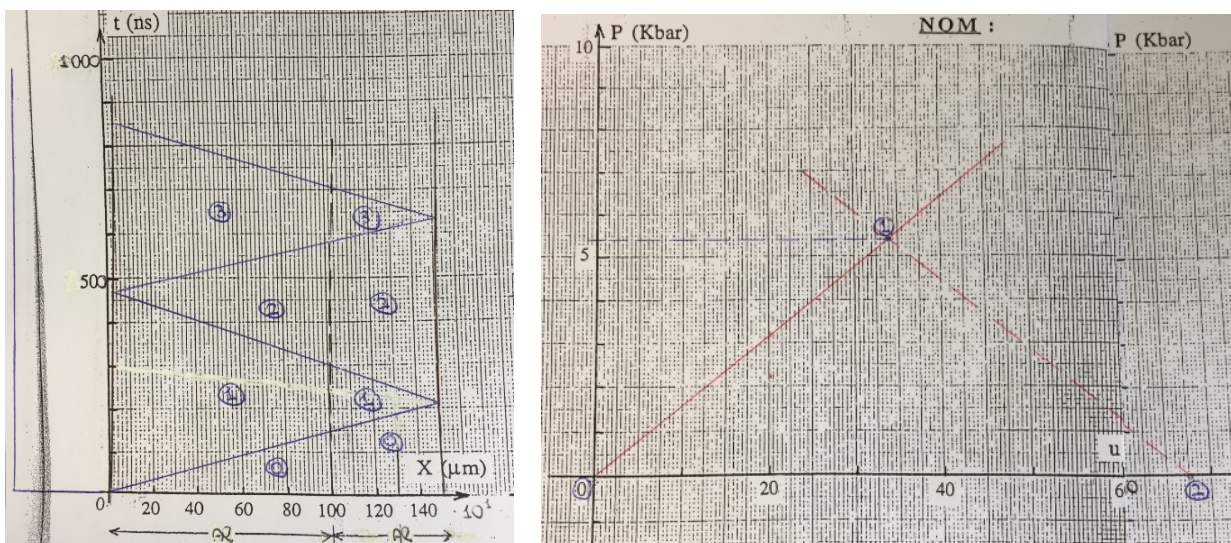


(a) X-Time diagram          (b) Pressure vs Velocity diagram

Figure 31: Analytical diagrams in Al-Al case (Uniform loading)

The main elements and nodes of our geometry are taken and specified in the legend. Placing the origin on the central axis in the Al part, these are the lower ("Extreme Al element") and the middle ("Middle Al element") elements of the Al layer, the elements at both sides of the interface plate-coating ("Interface Al element", "Interface Al coating/Epoxy element") and the last ones are in the middle ("Middle Al coating/Epoxy element") and at the extreme ("Extreme Al coating/Epoxy element") of the coating:
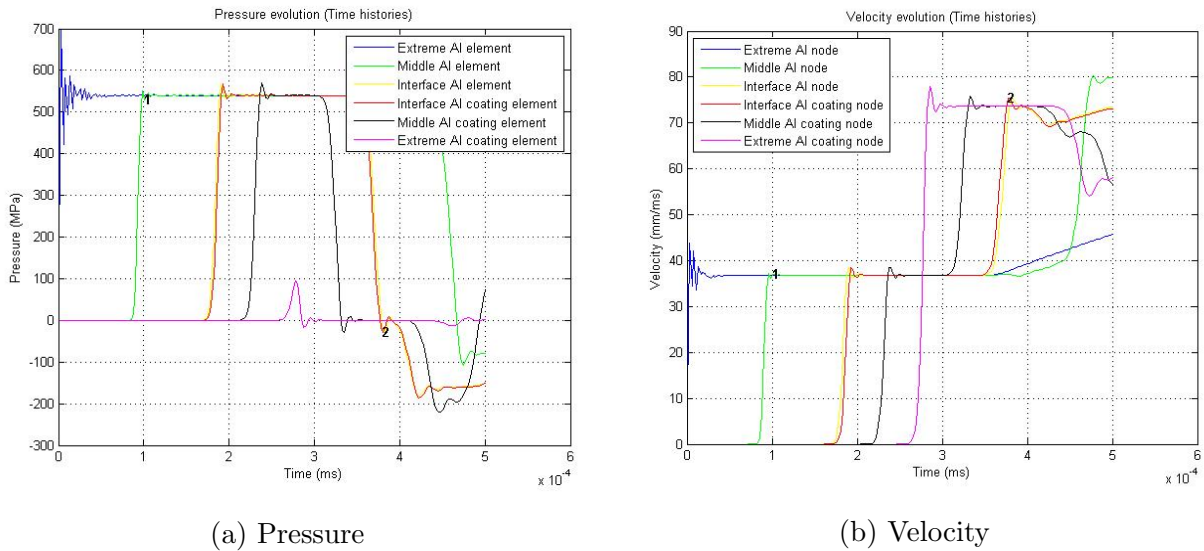
(a) Pressure

(b) Velocity

Figure 32: Time histories of the main elements in Al-Al case (Uniform loading)

Having a first quick look, all the elements for the Al-Al case achieve the complete amplitude set in the pre-processing, i.e a pressure around 540 Mpa and a velocity of 37 mm/ms which correspond to state 1, except for the last element in magenta corresponding to the extreme of the coating which tend to grow due to the brief crossing by the state 1, but it ends like the rest (except for the first element in the extreme of Al layer, which keeps the values of state 1) in state 2 whose region is next to the free surface at zero pressure and 74 mm/ms of velocity.

For the case of Al-Epoxy, the layout of the coating produces the reflection of the wave sooner, what induces the appearance of new states of pressure an velocity. The new state 2 remains in the Hugoniot curve of Epoxy at 200 Mpa of pressure and 60 mm/ms of velocity. Just to remark more characteristics that are perfectly seen, we have to mention that the elements at the interface (red and yellow) and in middle of both the plate and the coating (green and black, respectively) pass through that state 2.



(a) X-Time diagram

(b) Pressure vs Velocity diagram
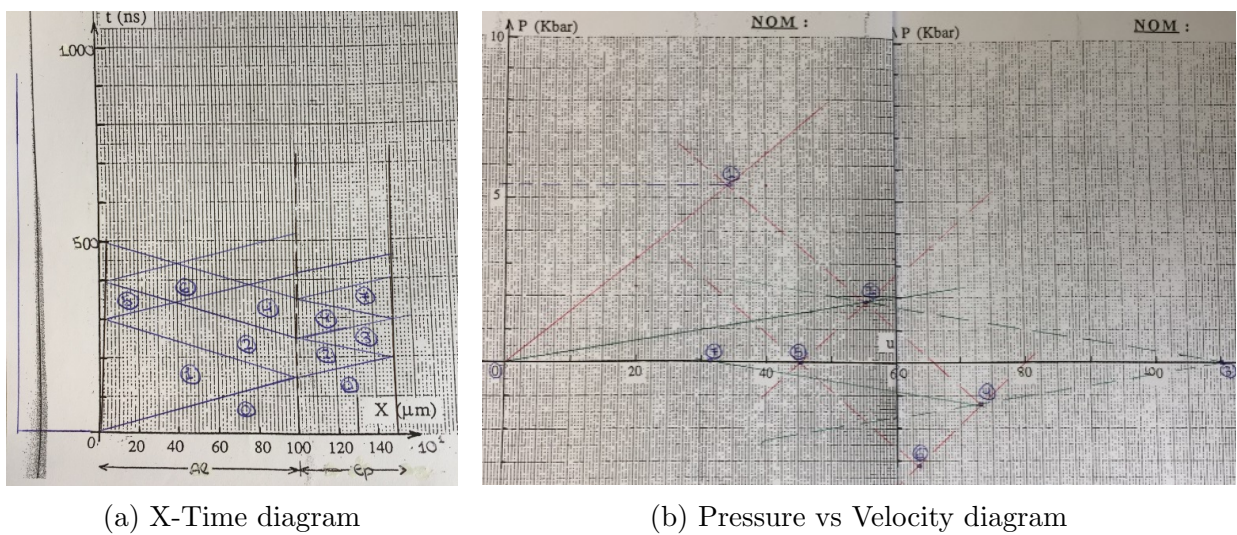
Figure 33: Analytical diagrams in Al-Epoxy case (Uniform loading)

Following the X-Time diagram, we can deduce that each element follows a different way passing trough different states which are created due to the higher quantity of reflections that happen. Now, there are state 3, 4 and 5 which are also affected by the 2D effects, what can make our theoretical diagram inconsistent. Only the state 3 which correspond to the release wave into the free surface, what doubles the velocity to 120 mm/ms at zero pressure, can be validated for the element at the extreme of the coating in magenta.

(a) Pressure
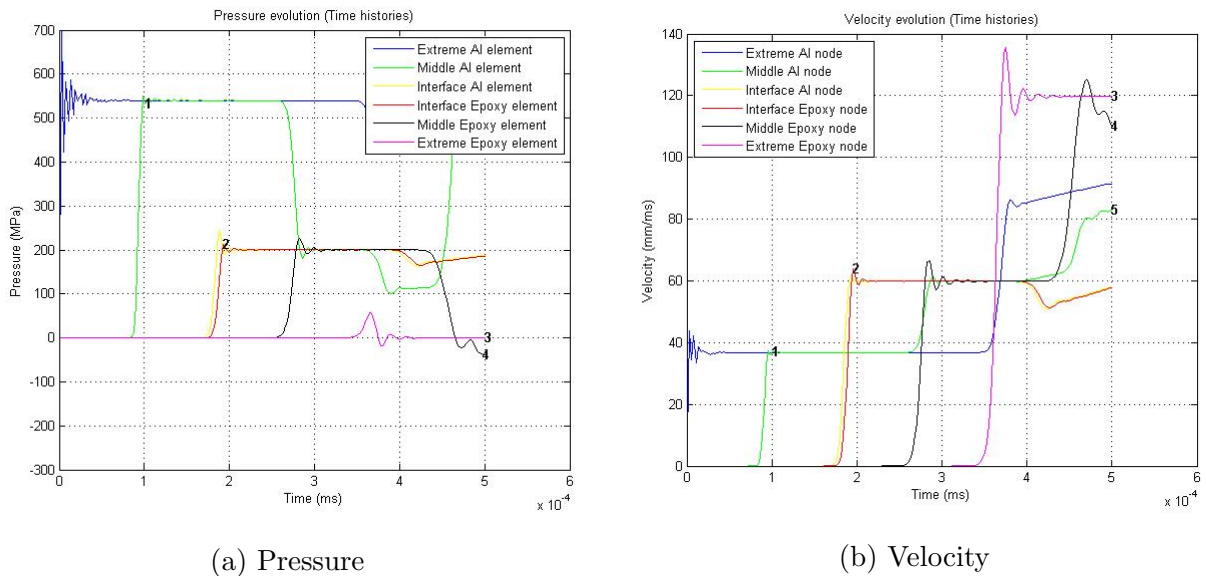
(b) Velocity

Figure 34: Time histories of the main elements in Al-Epoxy case (Uniform loading)

The meaning of the double velocity remains in the fact that the velocity of the plate free surface (which can be measured experimentally) is supposed to be the double of the wave propagation velocity of the material in the previous state coming from a wave release (drop in pressure). Generally, the evolution of the velocity of the plate free surface is directly related with the different states obtained in the "interior" part of the plate and it constitutes a consistent way of validation of the simullated models with the laser shock phenomena.

- **X-T diagram:**
  The arrival of the shock wave to the different interfaces which appear in its travel can display different behaviour. We expect that it passes through the Al-Al- interface without reflection until getting the back face, where reflection and the tensile stress are produced. For the case Al-Epoxy, the reflection happens in the proper interface avoiding that the wave reaches the back face and the corresponding tensile stresses appear. Everything can be observed in the X-T diagrams for each case:



(a) Al-Al
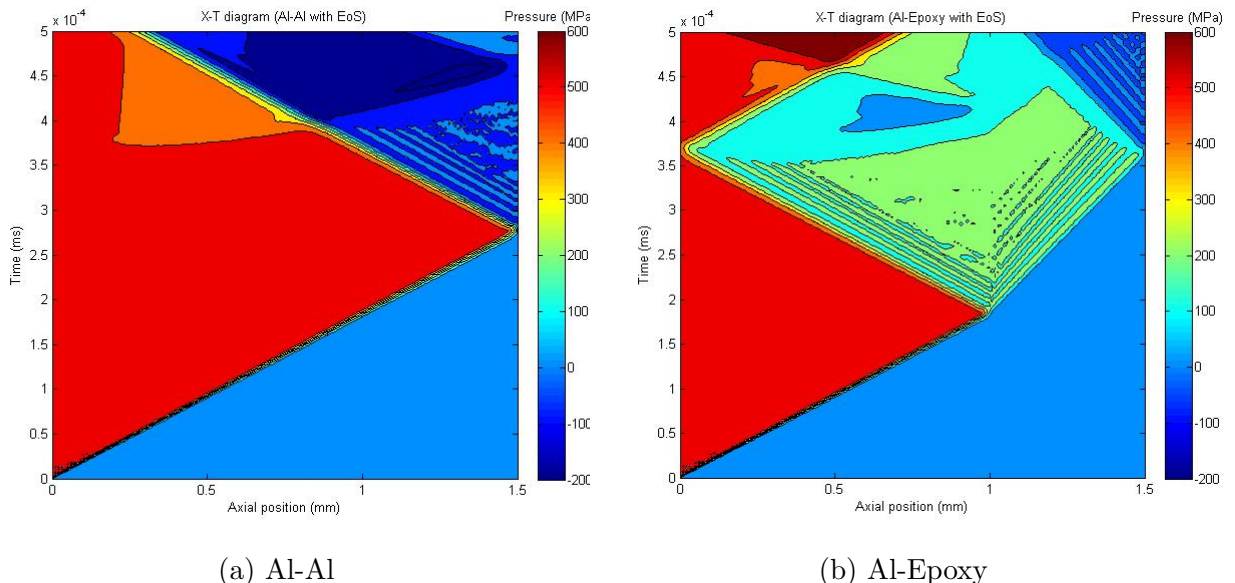
(b) Al-Epoxy

Figure 35: X-T diagrams for the different interfaces (Uniform loading)

We can observe that the tensile stresses (dark blue contours) are of higher magnitude in the case Al-Al and they appear even sooner than in the Al-Epoxy case, which seems to relieve those effects keeping positive stress prolongated in time. It fits the aim of the Epoxy coating adhered by laser shock, which tries to protect the part against agressive stresses coming from the medium or against the friction generated by hard materials in contact. The laser shock is a non-destructive test to test the adherence of thin layers thanks to its brief application, which allows to reproduce the limit tensile conditions that can trigger into the separation of the parts.

29

### 3.3.3 Optimization of the case

For this case, the section of optimization consists on applying the corresponding pressure distribution of our initial problem and showing the results of both cases (Al-Al and Al-Epoxy) once we have validated the uniform amplitude case.

First of all, let us start with the contour plot animation in order to check if the wave travels without excessive distorsion induced by the discontinuity of the partition region . The limits are fixed to the same values inside the Contour plot options from the upper bar. We expect to get similar behaviour of the wave depending on the material of the coating like the uniform amplitude case:
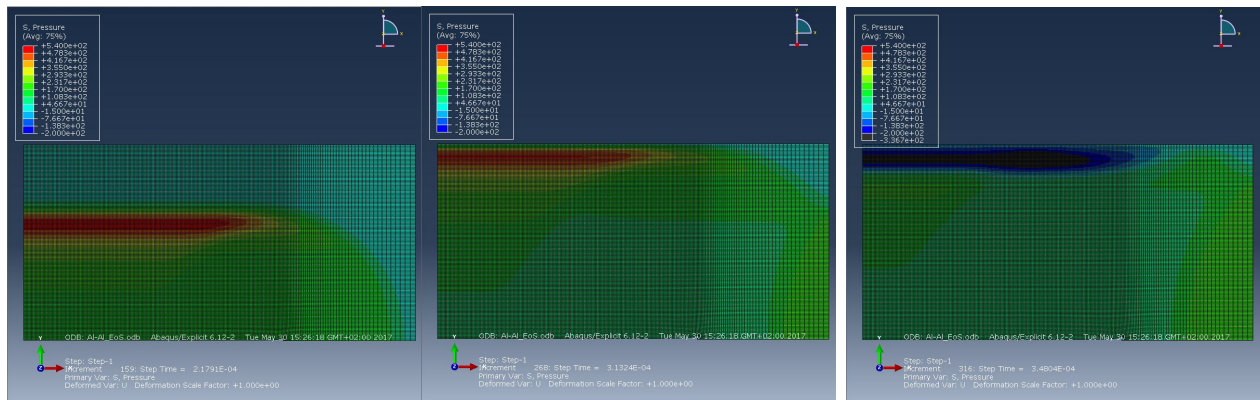


Figure 36: Pressure evolution in interface Al-Al with EoS
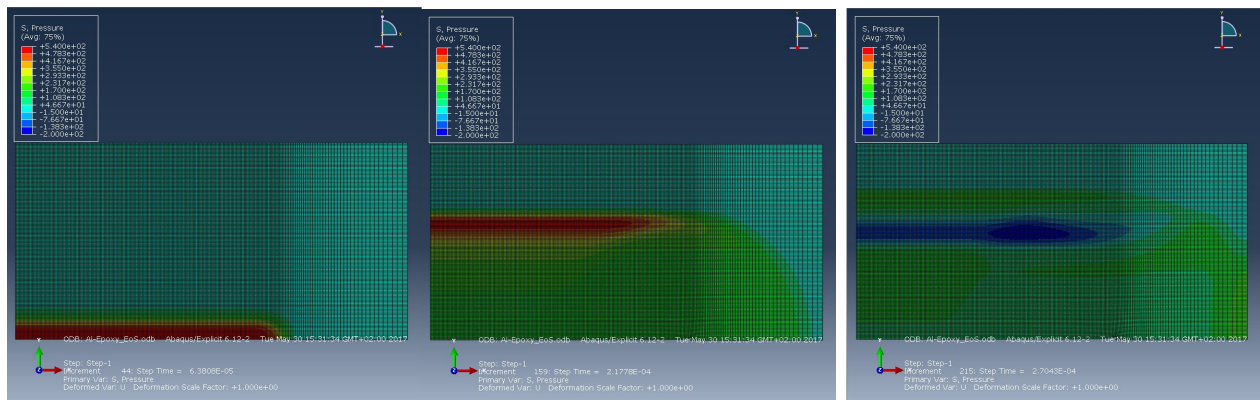


Figure 37: Pressure evolution in interface Al-Epoxy with EoS

The second step is showing the X-T diagram for each case, which can give us an idea of when the reflections are produced and the order of magnitude of induced tensile stresses. It allows to validate the use of the Epoxy layer for the adherence purpose explained previously. The arrival of the maximum pressure to the back face is avoided, what reduces the possibility of damage and failure, in addition to weaken the traction solicitations.

(a) Al-Al

(b) Al-Epoxy
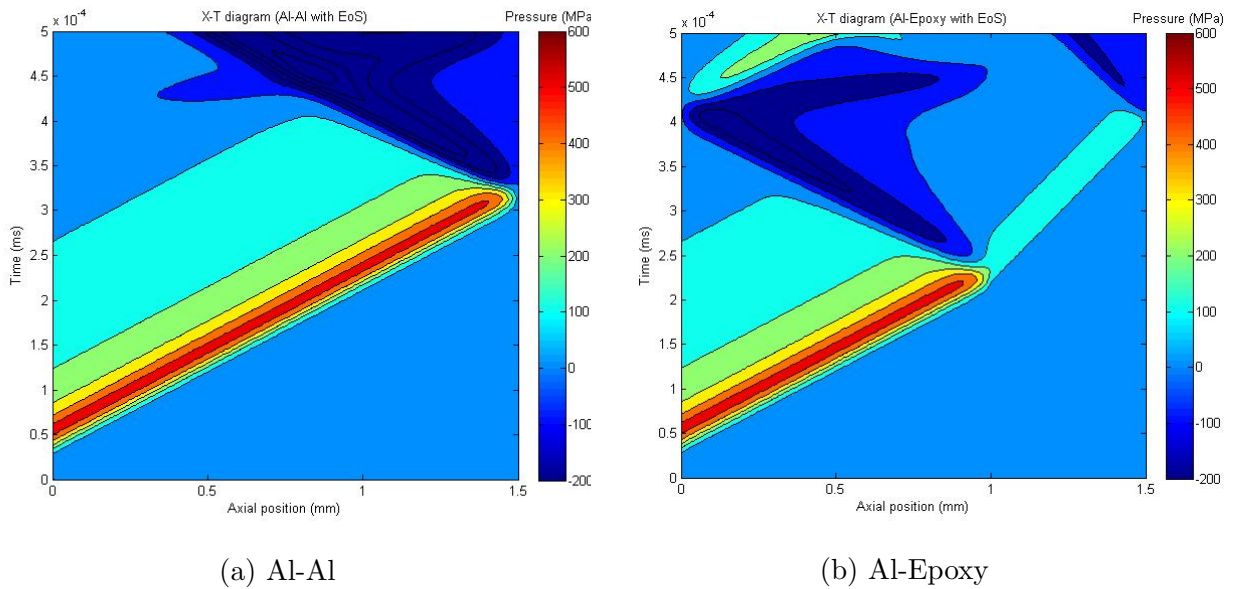
Figure 38: X-T diagrams for the different interfaces (Uniform loading)

### 3.3.4 Validation of the case

Finally, we have to compare the time histories with the analytical diagrams of P vs Time and X-Time. Let us start with the interface Al-Al which is simpler because of having no reflection at the interface plate-coating. The first step is building the analytical diagram for the case in order to have something to support our model.



(a) X-Time diagram

(b) Pressure vs Velocity diagram
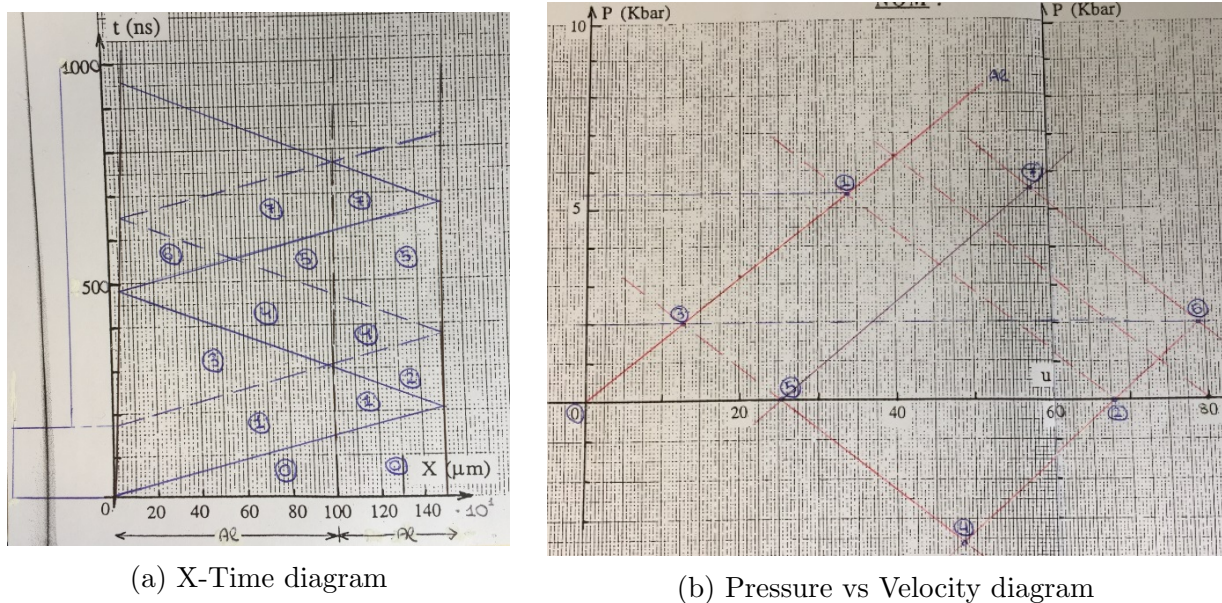
Figure 39: Analytical diagrams in Al-Al case

Here, we display the time histories of pressure and velocity taken from the output data text files at the main elements and nodes of the mesh, like the previous case:

(a) Pressure
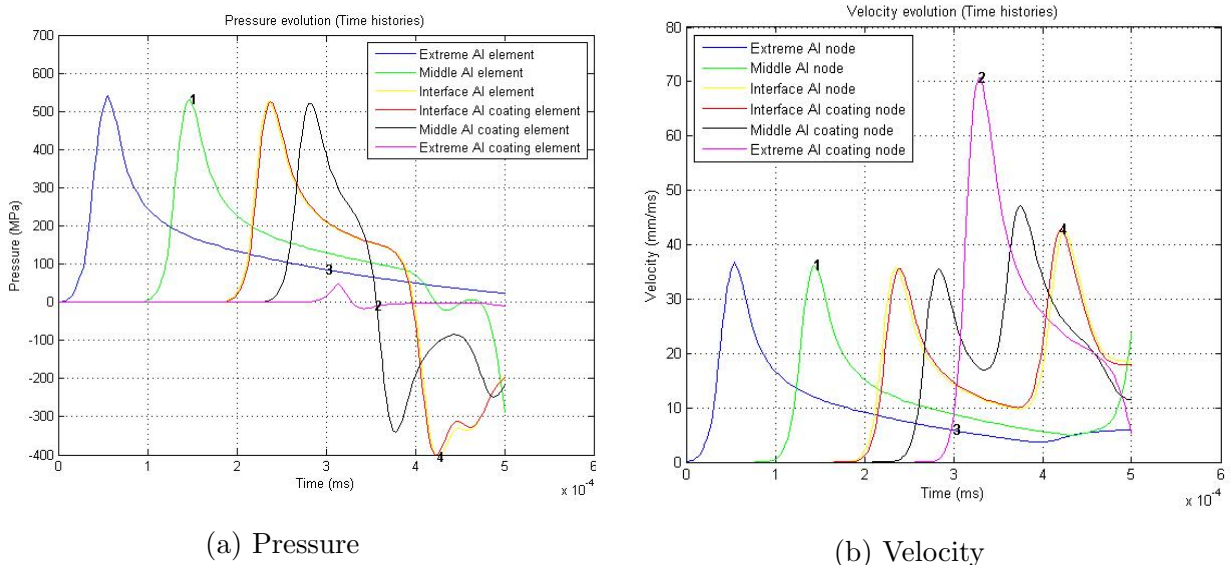
(b) Velocity

Figure 40: Time histories of the main elements in Al-Al case

For the elements in the Aluminum layer (blue in the extreme and green in the middle), they pass trough state 0 at zero pressure and velocity, state 1 at 540 MPa of pressure (maximum amplitude) and 35 mm/ms, state 3, whose validation is dependent on the point taken from the distribution as the release after the peak of pressure and whose conditions are not clearly specified at all in both plots. The green one tends towards state 4 at the end.

In the case of the elements at the interface (yellow and red plots), we can observe the entering in the state 4 with negative value of pressure (tensile stress) coming from a brief passing through state 3 and 2, respectively. The velocity of the wave at the state 4 is a bit higher than the state 1 as we can check.

For the elements of the coating (black in the middle and magenta in the extreme), there is a tendency to grow until the velocity of state 2, which should be the double of the state 1 for being the free surface velocity (explained in the previous cases), around 70 mm/ms. Then, the black one, coming from the first and passing through the second states briefly without reaching the maximum, ends at the negative pressure values of state 4, while the magenta one ends tending to zero pressure and lower velocity of state 5 after the peak.



(a) Pressure

(b) Velocity
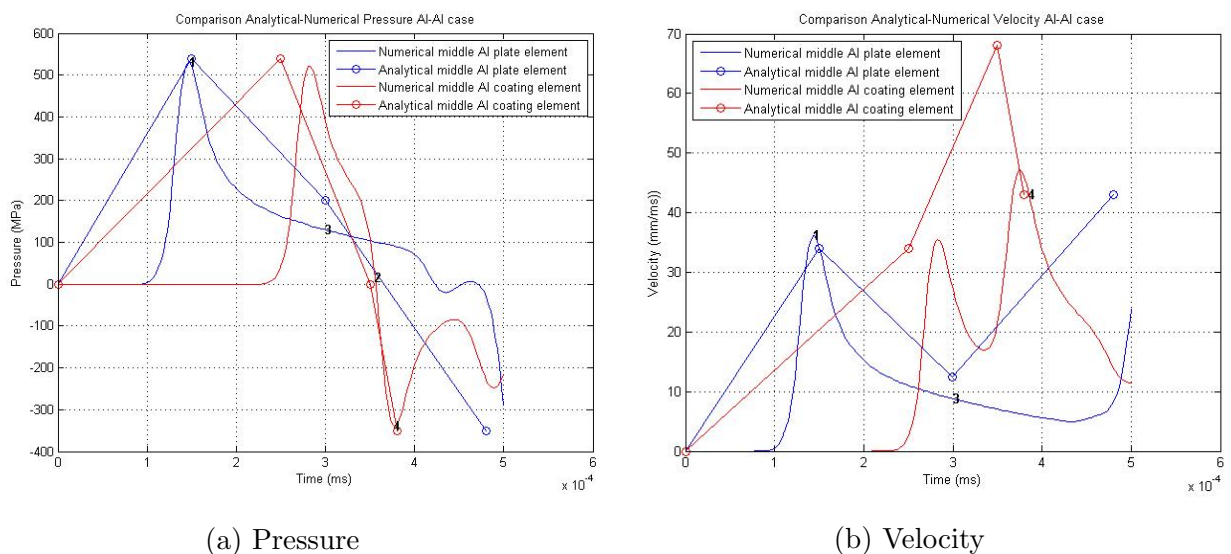
Figure 41: Analytical-numerical time histories for Al-Al case

Now, we proceed by the same way for the Al-Epoxy interface, i.e building the analytical diagrams for the validation of the case:

(a) X-Time diagram

(b) Pressure vs Velocity diagram
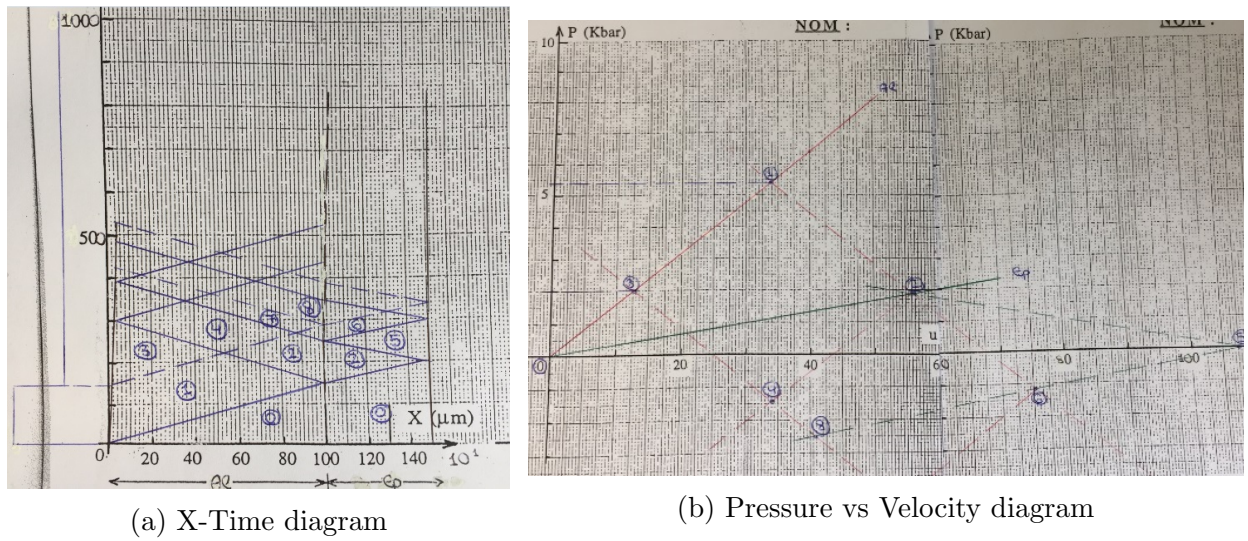
Figure 42: Analytical diagrams in Al-Epoxy case

Let us explain the differences found with respect to the previous case analyzing element by element. The element in the middle of the Al layer drops to the negative values of pressure at state 4 because its crossing through state 3 is very short, with a light tendence of recovering velocity.
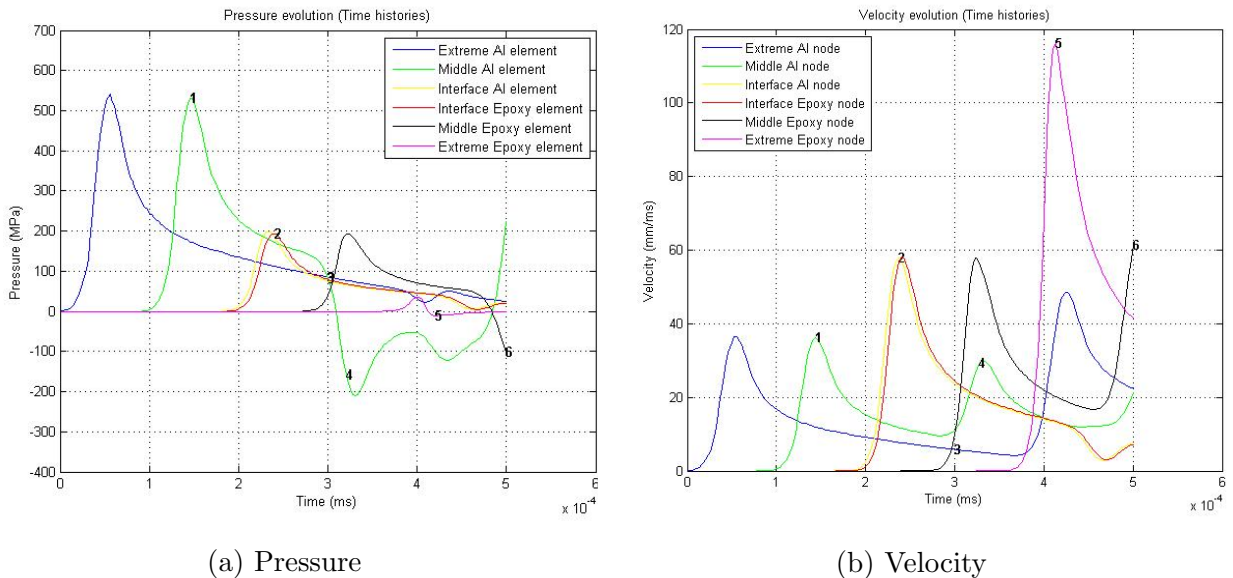


(a) Pressure

(b) Velocity

Figure 43: Time histories of the main elements in Al-Epoxy case

The elements at the interface (one of Al, another of Epoxy) does not cross state 1 practically and pass through a new state 2 at around 200 Mpa of pressure and 58 mm/ms of velocity. Then, they drop as a consequence of the pressure release towards states 6 and 5, respectively.This state 5 would correspond to the previous state 2 in the Al-Al case, at zero pressure and with the double of velocity of the incoming state 2 (around 116 mm/ms) and it is reached by the elements in the coating. The black in the middile only tends to grow in velocity, but drops in pressure after passign trough state 2, and the magenta in the extreme keeps closely at zero pressure and achieves the maximum velocity.

(a) Pressure          (b) Velocity
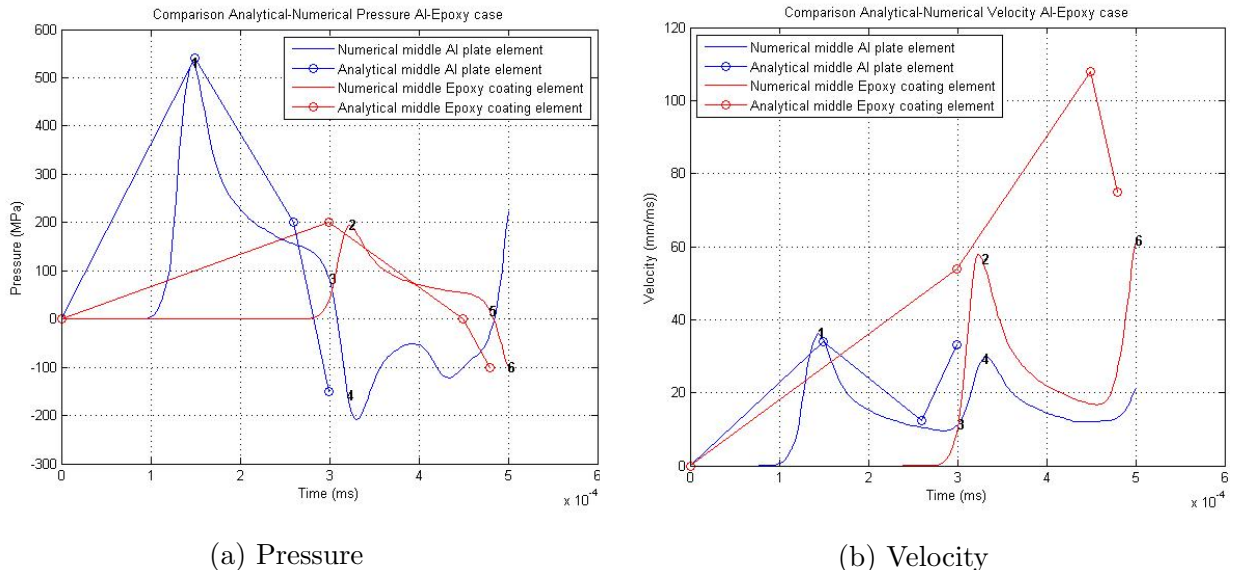
Figure 44: Analytical-numerical time histories for Al-Epoxy case

From a certain time instant in advance, the behaviour of the wave changes due to the appearance of 2D effects, as it has been explained previously. However, in spite of being an approximated analysis, it is enough to consider our model validated always having in mind the possibility to improve and extend the validation with a better reproduction of the pressure distribution building the theoretical diagrams and the study of those effects.

### 3.3.5 Addition of a glass layer to the flat plate-coating problem

This case has the aim of detecting the variation in stress solicitation which a certain geometry can suffer, by adding a layer of some kind of glass whose objective is producing a game of colours depending on the pressure levels. The objective is also improving the behaviour law of the substract and validate the equation of state of epoxy for the coating material, as well as testing the adherence like the previous case.

The new geometry is obtained by modifying the previous model with 1mm thick flat plate of Aluminum and 0.5 mm thick coating of Epoxy, by another one composed by 3 parts or instances: 1mm thick flat plate of Aluminum, 0.04 mm thick coating of Epoxy and 1mm thick flat plate of Glass. The new material Glass has to be added to our Property module with the following characteristics:

| **Material** | $\rho_0(kg/cm^3)$ | $c_0(mm/ms)$ | $s(-)$ | $\gamma(-)$ |
|:---:|:---:|:---:|:---:|:---:|
| Glass | 2.5 | 5.25 | $-2.2$ | 2 |

Table 4: Glass properties [4]

It is essential to keep in mind that glass has a specific behaviour which is not regular in the polar Hugoniot's curve due to its negative value. However, we have tried to adapt the case and display a simulation as accurated as possible. Since the negative value of the constant "s" is not admissible by Abaqus, it is set to 0 and the corresponding section is created and assigned to the added layer.

The way to glue the parts is the same than the previous final case, i.e by merging the nodes which are in contact in the corresponding command instead of setting a tie constraint for the surfaces in conflict. In addition, a few modifications are made concerning the constraints: now the upper/outer surface of the glass plate is completely fixed (all DoF) by an encastre. The rest, what means the right side of each part, are kept with the displacements in both axis constrainted, as well as the extra condition over the lower right side in the Al layer in order to reduce the deformation due to the discontinuity. For the load module, we apply the same pressure distribution taken from a datafile to the same portion of the lower side with a maximum amplitude of 540 MPa.

Meshing is performed differently for the three instances in order to get a optimization of the results. That means increasing the size of the Aluminum and glass elements vertically in order to be able to analyze the conservation by measuring the velocity at the same time instant depending on the propagation velocity of the different materials. The elements in the Epoxy coating are left with a size of 0.01 mm. From the partition until the extreme, a bias is also applied for every part to get a geometric increase of the size laterally till 0.04 mm in order to save some computational cost. After configuring all the steps of pre-processing, the case is ready to be submitted with a job:
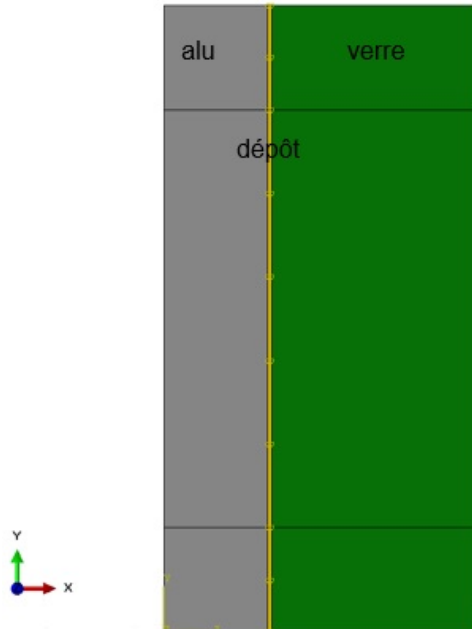


Figure 45: Flat plate case with glass layer [4]

Afterwards, we proceed to show the results and try to validate the model by the same way as previously. First of all, the contours animation following the levels of pressure travelling with the wave propagation. Now, there are more conflict zones because of the interfaces between the different materials, so we expect a different behaviour depending on where the wave is.

The range of contours is fixed between -200 and 540 MPa so that the wave propagation is perfectly followed. The impact of the wave to the Epoxy coating (1st row, 1st picture) produces the reflection in the way of tensile stress travelling downwards. At the same moment, some lower pressure is transferred through the coating to the Glass layer (1st row, 2nd picture). Then, we can obserbe the sooner arrival of the wave to the front face in the Al layer, as the propation velcotity is higher than in the glass layer (2nd row, 1st picture). Upwards, the wave gets the back face, the end of the geometry, a bit later (2nd row, 2nd picture) while the reflection has already been produced in the front face having positive stress in this region again.
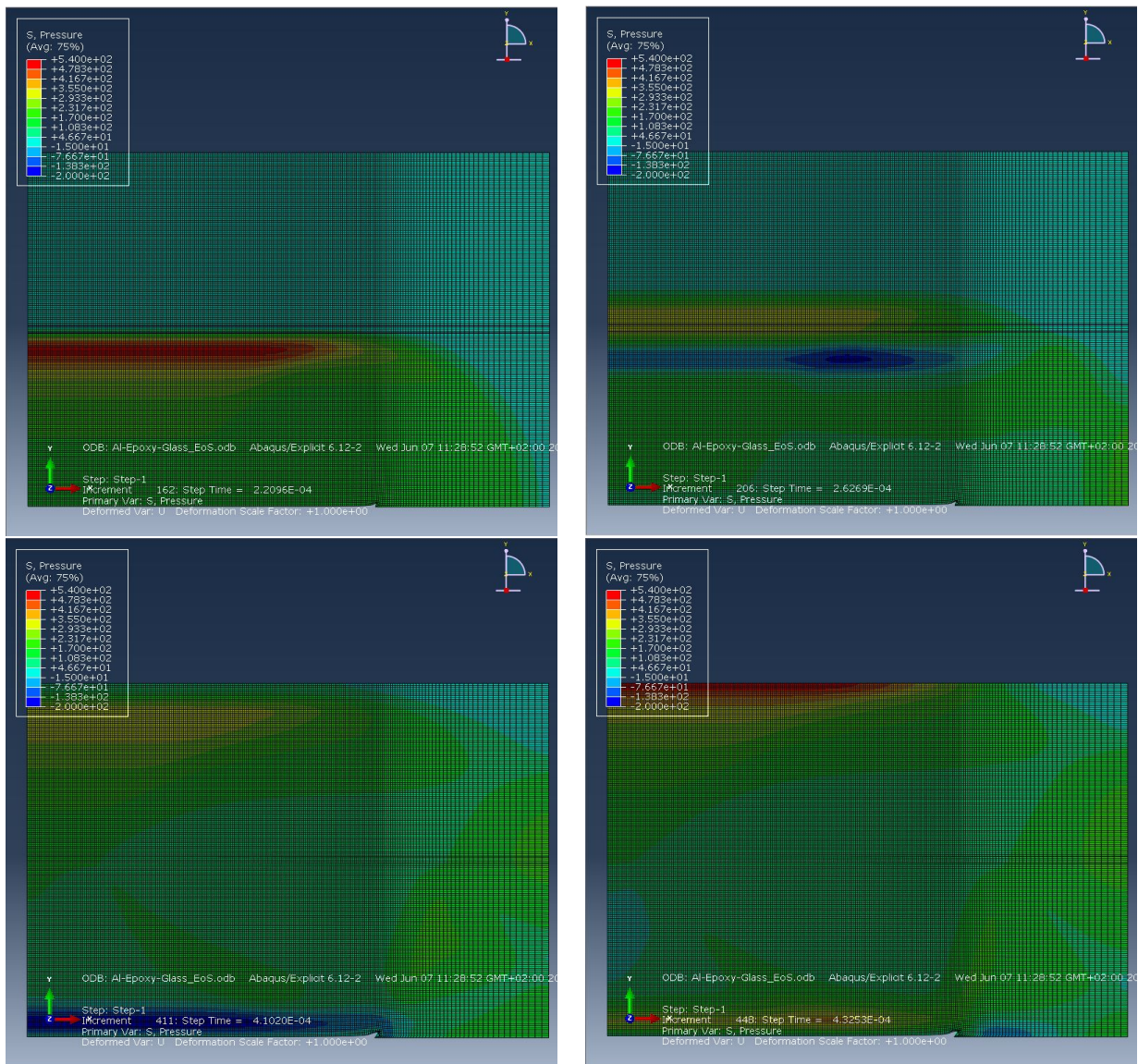
Figure 46: Evolution of pressure through the Al-Epoxy-Glass interfaces

The X-Time diagram shows the change in pressure when the wave achieves the interface regions and the corresponding time instant when it happens. Here, the tensile stess induced by the reflection in the Epoxy coating is perfectly distinguished between x=1 mm and x=1.04 mm and its propagation startes at approximately 2.5 ms. In addition, the arrival of the resultant wave after the coating to the back face of glass plate:
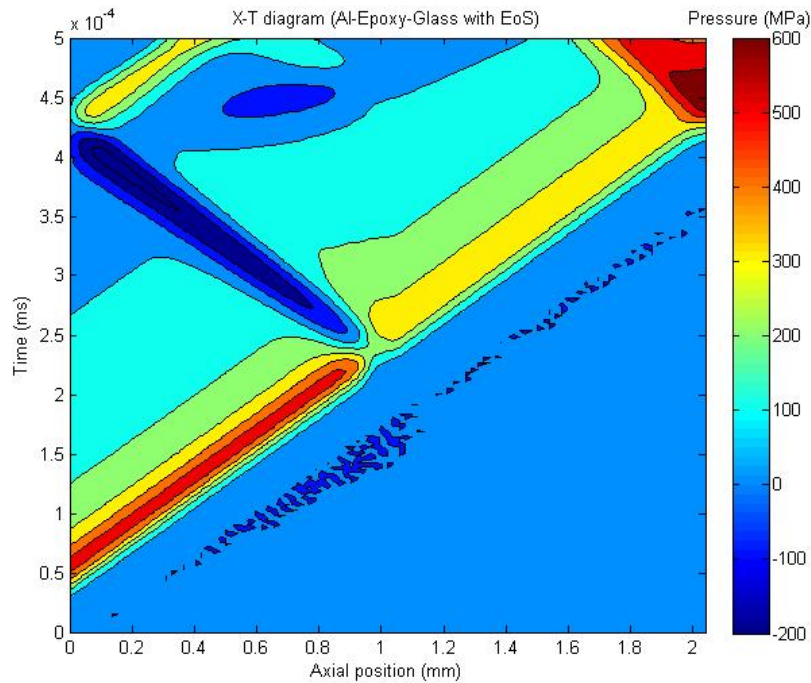
Figure 47: X-Time diagram of Al-Epoxy-Glass case

Finally, we take some time histories of the main elements and nodes of the geometry in order to study the transition of the wave through the interfaces. In this case, the objective is only showing that an accurated model for the addition of glass is possible, but the complete validation of the case cannot be approached by the theory due to the non-regular behaviour of Glass. The study of the wave propagation in such kind of material is proposed for further research work.

For the elements in the Aluminum layer (blue in the extreme and green in the middle), they pass trough state 0 at zero pressure and velocity, state 1 at 540 MPa of pressure (maximum amplitude) and 36 mm/ms, state 5, which is dependent on the point taken from the distribution as the release after the peak of pressure and both the pressure and velocity are lower than state 1. The blue one passes enterely by state 5 and finally gets state 7 at zero pressure but higher velocity than 1, what explains the final tendency to grow, while the green drops to state 6 which corresponds to tensile stresses (negative values of pressure) with another increase of velocity after the brief crossing through state 5.
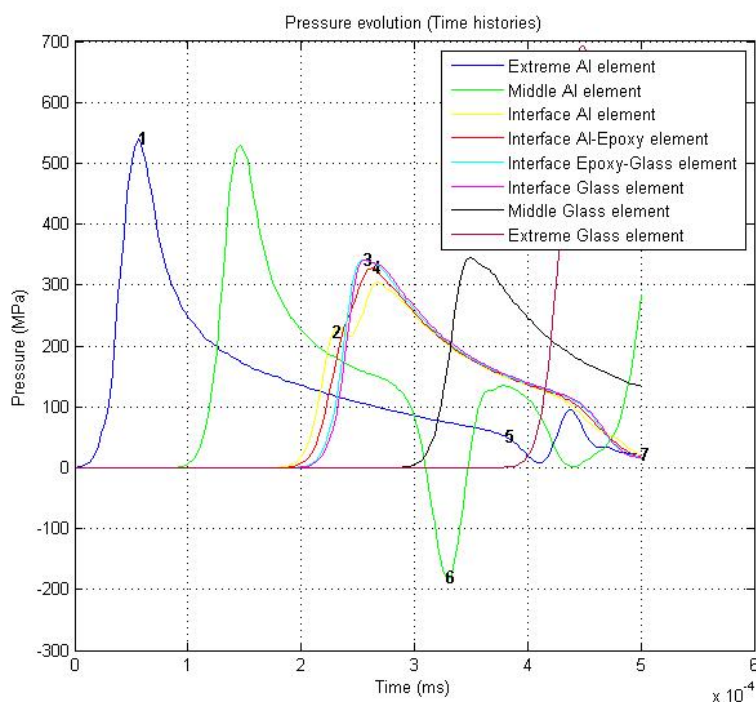


Figure 48: Pressure vs Time history of Al-Epoxy-Glass case

In the case of the elements at the first interface Al-Epoxy (yellow and red plots), we can observe both entering in the state 2 over the Hugoniot of Epoxy with a pressure around 200 MPa with the highest velocity (around 56 mm/ms) and continue growing above 300 MPa corresponding to state 4. For the second interface Epoxy-Glass (cian and magenta elements), they achieve state 3 before going to state 4 (both at similar pressures), but with a remarkable lower velocity around 26 mm/ms comparing with state 2. Then, they tend to recover slightly with state 9, but the end of the step does not allow to visualize it.
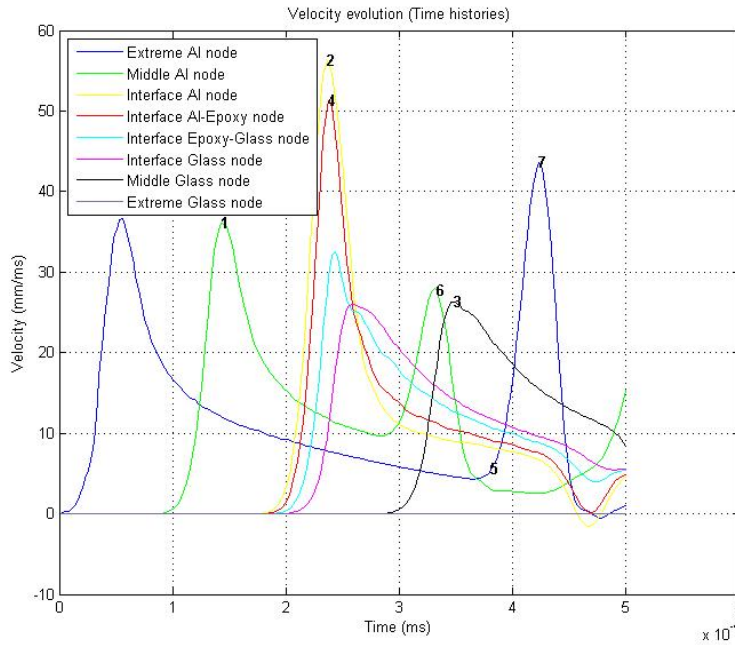


Figure 49: Velocity vs Time history of Al-Epoxy-Glass case

In the last instance, the elements of the Glass layer are the black in the middle and brown/grey in the extreme. The first one has the same evolution than the second interface, state 0, state 3 and state 9, without displaying this last one very well. The last one remains at zero pressure and it should double the velocity of state 3 when passing through state 10 which corresponds to the free surface velocity.

Although we have developed a quite accurated model, the generalities for compressive waves steepening is no longer valid for the case of the addition of the glass. The last picture corresponds to an amplification of the pressure time history for the elements at the interfaces, whose maximum value is identified around 330 MPa:
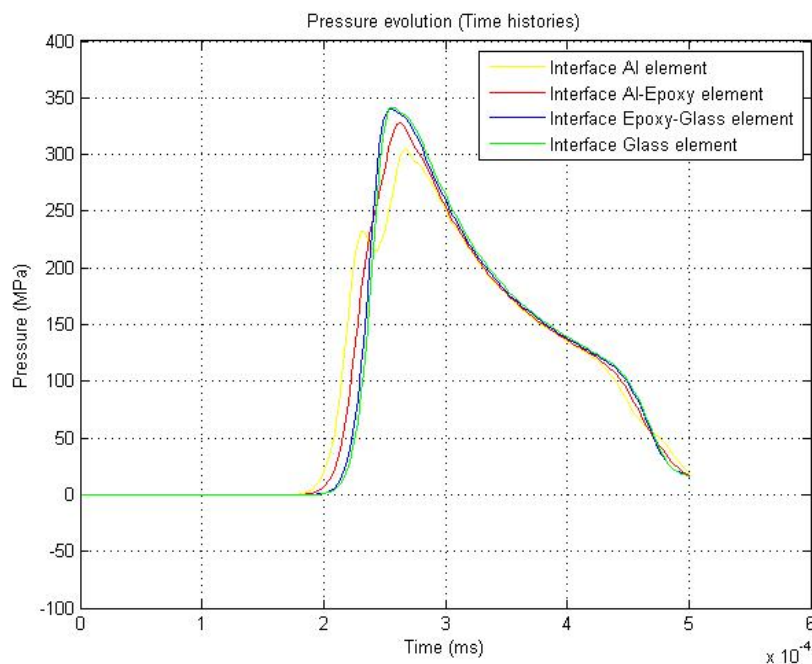


Figure 50: Maximum pressure at the coating

# 4 Conclusion

The numerical simulation can provide a better understanding of the key parameters acting on the response of a material or assemblies to shock loading. Several models have been developed to enhance the understanding of laser shock propagation in our cases. The aims of them are different, but the principle is the same: the propagation of a shock wave induced by a sudden appplication of an effort. In addition, it can predict the levels of stress-strain undertaken by materials under such loadings and help designing their survvivability, durability...

For achieving such objective, a simulation has to present a high level of liability. Therefore, for most of the cases studied we performed a comparison between the expected theoretical results and numerical ones. This helped very much in solving many possible mistakes whose effects cannot be easily found without such approach. However, the theory cannot approached the appearance of 2D effects as soon as the wave is propagated, what is perfectly distinguished in our simulations. Consequently, it is restricted to simple geometries (flat plates) with specific boundary conditions and ways of load application.

In the case of the projectile impact, the simulation gets the conditions which would induce damage in the back face of the plate by means of spalling, so we have got an approximated idea about the order of magnitude of the test conditions. The same happens with the case of the exposure of a flat plate with coatings to a pressure distribution. At first, testing the adherence of Epoxy to Aluminum by laser shock in order to test the bounding strenghts between both materials (behaviour laws) and then, the forward addition of other layers depending on the purpose/application, such as the glass plate of our final case.

Different ways to improve the simulations can be proposed, but at this moment we are able to make them even more complex adding the presence of recent discovered materials with other purposes, such as composites or certain fibers, whose modelization requires more detailed settings in the software.

The flexibility of working with a FEM software allows to change some parameters of the pre-processing in order to simulate different conditions of the problem, or even simpler, testing the behaviour of different materials in such case. Then, it is possible to extrapolate the numerical problem in order to apply in a real case once it has been validated with theory (the most reliable way), being equally successful. However, it does not stop being an approximated solution which cannot take into account all the conditions that are put in place in reality.

The work has been performed mainly for proving the applicability to Abaqus to cover the laser shocks modelling as well as RADIOSS, commonly used for this specific fast dynamic phenomena. From this point, the objective is expanding the modelization of the laser shock wave propagation phenomena to other more efficient numerical methods computationally speaking, such as Proper Generalized Decomposition Method (PGD), because it can be useful in terms of saving computational time and storage memory.

# 5 Annexes

## 5.1 Matlab script for spatial magnitudes (Displacement, Velocity)

```matlab
clear all; clc
%Open the text file
fid = fopen('Al-Epoxy_EoS_Nodes.txt','r');
tline = fgetl(fid);
i=0;
disp(tline)
%Reading line by line and storage in unique matrix ...x2
while ischar(tline)
    scnot=strfind(tline,'E-');
    if ~isempty(scnot)
    i=i+1;
    M(i,:)=str2num(tline);
    end
    if ~isletter(tline)
    i=i+1;
    M(i,:)=str2num(tline);
    end
    tline = fgetl(fid);
    disp(tline)
end
fclose(fid);
nSteps=201; %Spaced time intervals for reporting data
nVariables=2; %Field output variables selected for XY data in Abaqus
nNodes=151; %If not merged, change to 152
%nNodes=108; %For case Al-Epoxy-Glass

%Separation in general vectors time/spatial
for j=1:nVariables
    var_start=(j-1)*nSteps*nNodes+1;
    var_end=j*nSteps*nNodes;
    time(:,j)=M(var_start:var_end,1);
    spatial(:,j)=M(var_start:var_end,2);
end
%Building the data matrix for each variable of every node along the time
%intervals
for i=1:nNodes
    nd_start=(i-1)*nSteps+1;
    nd_end=i*nSteps;
    T(i,:)=time(nd_start:nd_end,1);
    displacement(i,:)=spatial(nd_start:nd_end,1);
    velocity(i,:)=spatial(nd_start:nd_end,2);
end
%Reorganisation of the matrices in order to read by rows the value of the
%corresponding magnitude in every node at the same time instant
[m,n]=size(velocity);
for k=1:m
    U(:,k)=displacement(k,1:n);
    V(:,k)=velocity(k,1:n);
    Yaxis(:,k)=T(k,1:n);
end

%Plotting time histories of Velocity for the main nodes of the mesh (Al-Al
%and Al-Epoxy cases)
figure
plot(Yaxis(:,1),V(:,1),'b')
hold on
plot(Yaxis(:,50),V(:,50),'g')
hold on
plot(Yaxis(:,100),V(:,100),'y')
hold on
plot(Yaxis(:,102),V(:,102),'r')
hold on
plot(Yaxis(:,126),V(:,126),'k')
hold on
plot(Yaxis(:,151),V(:,151),'m')
title('Velocity evolution (Time histories)')
ylabel('Velocity (mm/ms)')
xlabel('Time (ms)')
legend('Extreme Al node','Middle Al node','Interface Al node','Interface Epoxy node','
%legend('Extreme Al node','Middle Al node','Interface Al node','Interface Al coating n
grid on

%Plotting time histories of Velocity for the main nodes of the mesh
```

```
%(Al-Epoxy-Glass case)
% figure
% plot(Yaxis(:,1),V(:,1),'b')
% hold on
% plot(Yaxis(:,26),V(:,26),'g')
% hold on
% plot(Yaxis(:,51),V(:,51),'y')
% hold on
% plot(Yaxis(:,53),V(:,53),'r')
% hold on
% plot(Yaxis(:,55),V(:,55),'c')
% hold on
% plot(Yaxis(:,57),V(:,57),'m')
% hold on
% plot(Yaxis(:,81),V(:,81),'k')
% hold on
% plot(Yaxis(:,108),V(:,108),'color',rand(1,3))
% hold on
% title('Velocity evolution (Time histories)')
% ylabel('Velocity (mm/ms)')
% xlabel('Time (ms)')
% legend('Extreme Al node','Middle Al node','Interface Al node','Interface Al-Epoxy no
% grid on
```

## 5.2 Matlab script for stresses ($\sigma_{yy}$, Pressure)

```
clear all; clc;
%Open the text file
fid = fopen('Al-Epoxy_EoS.txt','r');
tline = fgetl(fid);
i=0;
disp(tline)
%Reading line by line and storage in unique matrix ...x2
while ischar(tline)
    Eplus=strfind(tline,'E+');
    Eminus=strfind(tline,'E-');
    if (isempty(Eplus)) (isempty(Eminus))
    i=i+1;
    M(i,:)=str2num(tline);
    end
    if ~isletter(tline)
    i=i+1;
    M(i,:)=str2num(tline);
    end
    tline = fgetl(fid);
    disp(tline)
end
fclose(fid);
nSteps=201; %Spaced time intervals for reporting data
nVariables=2; %Field output variables selected for XY data in Abaqus
nElements=150;
%nElements=107; %For case Al-Epoxy-Glass

%Separation in general vectors time/stress
for j=1:nVariables
    var_start=(j-1)*nSteps*nElements+1;
    var_end=j*nSteps*nElements;
    time(:,j)=M(var_start:var_end,1);
    stress(:,j)=M(var_start:var_end,2);
end
%Building the data matrix for each variable of every element along the time
%intervals
for i=1:nElements
    el_start=(i-1)*nSteps+1;
    el_end=i*nSteps;
    T(i,:)=time(el_start:el_end,1);
    pressure(i,:)=stress(el_start:el_end,1);
    Syy(i,:)=stress(el_start:el_end,2);
end
%Reorganisation of the matrices in order to read by rows the value of the
%corresponding magnitude in every element at the same time instant
[m,n]=size(pressure);
for k=1:m
    P(:,k)=pressure(k,1:n);
    S22(:,k)=Syy(k,1:n);
    Yaxis(:,k)=T(k,1:n);
end
%Division of the line of elements in equally spaced sections (nElements)
```

```
SpaceVar=linspace(0,1.5,m);
% SpaceVar=linspace(0,2.04,m); %For case Al-Epoxy-Glass
for l=1:n
    Xaxis(l,:)=SpaceVar;
end

%Space vs Time diagram with contours of pressure
figure
contourf(Xaxis,Yaxis,P)
colormap jet
h=colorbar;
set(get(h,'title'),'string','Pressure_(MPa)')%,'Rotation',90.0,'Position',[2 2.5E-04 0
title('X-T_diagram_(Al-Epoxy_with_EoS)')
%title('X-T diagram (Al-Al with EoS)')
%title('X-T diagram (Al-Epoxy-Glass with EoS)')
xlabel('Axial_position_(mm)')
ylabel('Time_(ms)')
caxis([-200, 600])
axis([0 1.5 0 5e-4])

% 3D plot of Space vs Time vs Pressure with rounded tensile stresses
% figure
% for i=1:25:m
% plot(Yaxis(:,i),P(:,i),'color',rand(1,3))
% hold on
% [tens, pos]=min(P(:,i));
% plot(Yaxis(pos,i),tens,'ro')
% hold on
% end
% plot(Yaxis(:,m),P(:,m),'color',rand(1,3))
% hold on
% [tens, pos]=min(P(:,m));
% plot(Yaxis(pos,m),tens,'ro')
% % plot3(Xaxis,Yaxis,P)
% % xlabel('Axial position (mm)')
% % ylabel('Time (ms)')
% % zlabel('Pressure (MPa)')
% title('Pressure evolution (Al-Epoxy with EoS)')
% xlabel('Time (ms)')
% ylabel('Pressure (MPa)')
% g=text(3E-04,-180,'Interface Tensile stress');
% set(g,'FontWeight','bold');
% grid on

%Plotting time histories of Pressure for the main elements of the mesh
%(Al-Al and Al-Epoxy cases)
figure
plot(Yaxis(:,1),P(:,1),'b')
hold on
plot(Yaxis(:,51),P(:,51),'g')
hold on
plot(Yaxis(:,100),P(:,100),'y')
hold on
plot(Yaxis(:,101),P(:,101),'r')
hold on
plot(Yaxis(:,125),P(:,125),'k')
hold on
plot(Yaxis(:,150),P(:,150),'m')
title('Pressure_evolution_(Time_histories)')
xlabel('Time_(ms)')
ylabel('Pressure_(MPa)')
grid on
legend('Extreme_Al_element','Middle_Al_element','Interface_Al_element','Interface_Epox
%legend('Extreme Al element','Middle Al element','Interface Al element','Interface Al
axis([0 6*10^-4 -300 700])

%Plotting time histories of Pressure for the main elements of the mesh
%(Al-Epoxy-Glass case)
% figure
% plot(Yaxis(:,1),P(:,1),'b')
% hold on
% plot(Yaxis(:,26),P(:,26),'g')
% hold on
% plot(Yaxis(:,51),P(:,51),'y')
% hold on
% plot(Yaxis(:,52),P(:,52),'r')
% hold on
% plot(Yaxis(:,55),P(:,55),'c')
% hold on
```

```matlab
% plot(Yaxis(:,56),P(:,56),'m')
% hold on
% plot(Yaxis(:,81),P(:,81),'k')
% hold on
% plot(Yaxis(:,107),P(:,107),'color',rand(1,3))
% hold on
% title('Pressure evolution (Time histories)')
% xlabel('Time (ms)')
% ylabel('Pressure (MPa)')
% grid on
% legend('Extreme Al element','Middle Al element','Interface Al element','Interface Al
% axis([0 6*10^-4 -300 700])

% Pressure time histories of the elements in the coating
% figure
% plot(Yaxis(:,101:104),P(:,101:104))
% title('Maximum Pressure coating')
% xlabel('Time (ms)')
% ylabel('Pressure (MPa)')
% grid on
% axis([2e-04 5e-4 0 400])
```

# List of Figures

# List of Tables

# References

[1] Michel Boustie, Laurent Berthe, T De Rességuier, and Michel Arrigoni. Laser shock waves: fundamentals and applications. In *Proc. 1st Int. Symp. On Laser Ultrasonics: Science, Technology and Applications*, 2008.

[2] Romain Ecault. *Etude expérimentale et numérique du comportement dynamique de composites aéronautiques sous choc laser. Optimisation du test d'adhérence par ondes de choc sur les assemblages composites collés.* PhD thesis, ISAE-ENSMA Ecole Nationale Supérieure de Mécanique et d'Aérotechique-Poitiers, 2013.

[3] Romain Ecault, Laurent Berthe, Michel Boustie, Fabienne Touchard, Emilien Lescoute, Arnaud Sollier, Patrick Mercier, and Jacky Benier. Observation of the shock wave propagation induced by a high-power laser irradiation into an epoxy material. *Journal of Physics D: Applied Physics*, 46(23):235501, 2013.

[4] F. Gehring. Modélisation abaqus/explicit des chocs laser pour l'anr chococomp. Technical report, Institut P' - ENSMA Département Physique et Mécanique des Matériaux - 86961 Futuroscope Chasseneuil Cedex, 2015.

[5] Jean-Paul CUQ-LELANDAIS Didier LOISON Michel BOUSTIE Thibaut DE RESSEGUIER. Utilisation de la suite hyperworks v 10.0 pour la modélisation de phénom'enes de dynamique rapide avec radioss - impact d'un projectile sur une plaque métallique a.4 - lois et param'etres matériaux. Technical report, ISAE-ENSMA École Nationale Supérieure de Mécanique et d'Aérotechnique - 86960 Futuroscope Cedex (France), 2010-2011.

[6] Hongbin Song. *Analyse expérimentale et numérique de la distribution des contraintes résiduelles induites par choc-laser dans les alliages d'aluminium.* PhD thesis, Arts et Métiers ParisTech, 2010.