Plate Perforation Analysis

Type of Solver: ABAQUS CAE/Explicit

Dynamic Analysis – Plate Perforation from Projectile Impact

Introduction:

Perforation of metallic plates during projectile impact is a complex process, commonly involving elastic and plastic deformation, strain and strain rate hardening effects, thermal softening, crack formation, adiabatic shearing, plugging, petalling and even shattering. These effects depend on the properties and geometries of projectile and target and on the relative velocity of the colliding bodies. In this practical class, you will model the perforation of a deformable metallic plate when struck by a rigid, spherical projectile over a range of impact velocities. You will run analyses assuming both rate dependent and rate-independent material behaviour, and compare your predictions.

Problem Description:

You have been asked by the Ministry of Defence to determine the ballistic limit (minimum impact velocity required to fully perforate a target) of 0.4 mm thick steel plates, impacted by spherical projectiles that are 8 mm in diameter. The projectiles have a mass of 0.002 kg. The steel mechanical properties have been measured for you. The plastic mechanical behaviour is well described by the Johnson and Cook phenomenological plasticity model.

Johnson-Cook Theory

The Johnson & Cook phenomenological plasticity relation defines the flow stress as a function of equivalent plastic strain, strain rate and temperature. The model is frequently used in impact analyses because of its simplicity. One other major benefit of this model is that the various phenomena such as strain hardening, strain rate hardening and temperature softening can be uncoupled. When only strain hardening and temperature softening are coupled, the effective stress is given by:

$$(1) \qquad \overline{\sigma}_d = \left[A + B(\overline{\varepsilon}_{pl})^n \right](1 - \hat{\theta}^m)$$

When strain rate hardening is deemed to be significant, then the dynamic flow stress is expressed by the following relation:

$$(2) \qquad \overline{\sigma}_d = \left[A + B(\overline{\varepsilon}_{pl})^n \right]\left[1 + C \ln\left(\frac{\dot{\overline{\varepsilon}}_{pl}}{\dot{\varepsilon}_0} \right) \right](1 - \hat{\theta}^m)$$

where σ_d is the dynamic flow stress, $\overline{\varepsilon}_{pl}$ is the equivalent plastic strain, $\dot{\overline{\varepsilon}}_{pl}$ is the equivalent plastic strain rate, $\dot{\varepsilon}_0$ is a reference strain rate, *A*, *B*, *n*, *m* and *C* are material parameters and $\hat{\theta}^m$ is the non-dimensional temperature. The constant *A* is the yield stress under quasi-static conditions, *B* and *n* are strain hardening parameters, *m* controls the temperature dependence and *C* the strain rate dependence.

(a) Build a 3-dimensional FE model. Since the plates are so thin, assume their geometry can be represented by a shell with an 80 mm diameter. Assume also that the projectile does not deform (either elastically or plastically). Beyond the elastic limit, the strain hardening behaviour of the plate is best described using the Johnson and Cook constitutive plasticity model (Eq.1) – recall that this equation neglects any strain rate hardening contribution. Fracture in the plates will be modelled using a very simple critical plastic strain fracture criterion. The Johnson and Cook material property data for Weldox 460E steel are given in Table I. Using this information, determine the ballistic limit of the steel plates when struck normal to their plane, using a measured (quasi-static) uniaxial plastic fracture strain of 0.33. Predict also the energy absorbed by the steel plates for 5 impact speeds between the ballistic limit and 600 m s^{-1}, when struck normal to their plane.

(b) Strain rate hardening effects were neglected in **(a)**. However, the steel plates are known to exhibit rate-hardening behaviour. Fortunately, the Johnson and Cook plasticity model can be modified to include the effects of strain rate hardening (Eq.2). Modify your existing material model to include the Johnson and Cook strain rate hardening parameters (Table I) and the strain rate-dependent fracture behaviour (Table II). Re-run your simulations, and predict new ballistic limit and the absorbed energy as a function of the impact speed once more. Compare these predictions with those from **(a)**. Comment on the significance of strain rate hardening in this particular steel alloy.

(c) (OPTIONAL) Determine the ballistic limit of the steel plates when impacted at 45°. Use the fully coupled form of the Johnson and Cook equation.

Material Property Data

Table I   Johnson and Cook Material Property Data

A	B	n	m	T_{melt}	T_{tran}	C	Ref.Strain rate
Mpa	Mpa			K	K		s^{-1}
310	1000	0.65	1	1673	293	0.07	0.01

Table II   Rate-dependent Fracture Data

Strain Rate s^{-1}	Stress Triaxiality	Fracture Strain
0.01	0.33	0.33
0.1	0.33	0.32
1	0.33	0.31
10	0.33	0.30
100	0.33	0.29
1000	0.33	0.28
10000	0.33	0.27

Table III

Density	Elastic Constants	
ρ	E	ν
kg m^{-3}	GPa	
7800	200	0.33

Solution (a):

- Start ABAQUS/CAE. At the **Start Session** dialogue box, click **Create Model Database.**

- From the main menu bar, select **Model→Create**. The **Edit Model Attributes** dialogue box appears; name the model `Plate Perforation`


A. MODULE→PART

Under the Part module, we will construct the plate and projectile.

1. From the main menu bar, select **Part→Create**

2. The **Create Part** Dialogue box appears. Name the part `Steel Plate` and fill in the options as shown in **Fig.A1**. Click Continue to create the part.

3. From the main menu bar, select **Add→Circle**

    (a) Select the co-ordinates (*0, 0)* for the centre of the circle in the prompt area**.**

    (b) Select the co-ordinates (0.04*, 0)* for the perimeter point

    (c) Click [X] and then **Done** in the prompt area.

**4.** You must now create two partitions. From the main menu bar
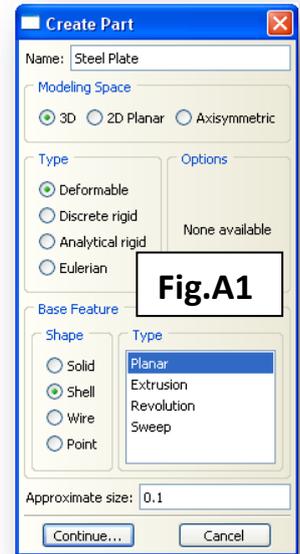
select **Tools→Partition**. The **Create Partition** dialogue box will open (**Fig.A2**).

    (a) Select **Face** as the **Type** and **Sketch** as the **Method**.

    (b) Using the mouse cursor select the edge of the part.

    (c) Using the **Create Circle** tool, create two circles; one with a diameter of 70 mm and the other with a diameter of 10 mm

    (d) Click [X] and then **Done** in the prompt area.
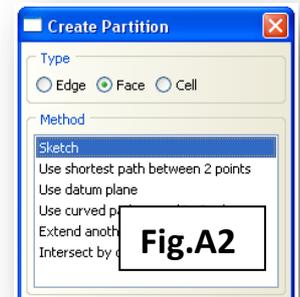
5. Now create an **Analytical Rigid** spherical projectile with an 8 mm diameter (**Fig.A3**). Once done, create a **Reference Point** at the centre. (**Tools→Reference Point**).

6. Create a set at the **Reference Point** and call it `Projectile Set.`

Reference Point

Fig.A3
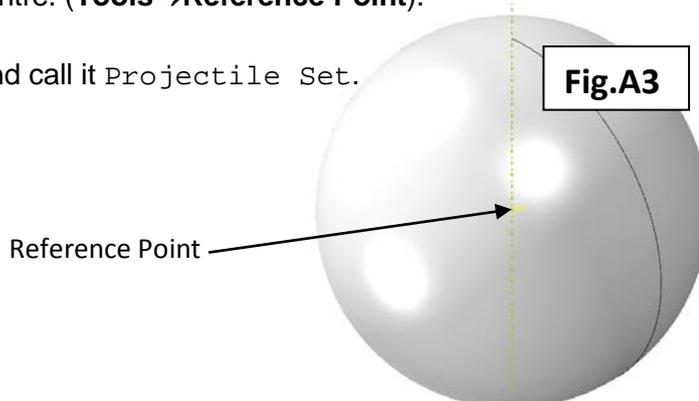
B. MODULE→PROPERTY

In this module (property), you will define the plate material properties. The strain hardening behaviour will be described using the Johnson and Cook plasticity relation. Fracture will be modelled by defining a critical plastic strain.

1. From the toolbox, select the **Create Material** tool. The Edit Material dialogue box will open.

    (a) Name the material `Weldox 460E`.

    (b) Define the density and elastic constants (Table III).

    (c) Select **Mechanical→Plasticity→Plastic** as shown in **Fig.B1**.

    (d) Under **Hardening**, select **Johnson-Cook** (**Fig.B2**). Fill in the data according to Table I.

    (e) To define the critical plastic strain at fracture, select **Mechanical→Damage for Ductile Metals→Ductile Damage** (**Fig.B3**). The quasi-static fracture strain is 0.33. The stress triaxiality is 0.33 and the reference strain rate is 0.01.

    (f) In the **Suboptions** drop-down menu, select **Damage Evolution** (**Fig.B4**). Choose a **Displacement at Failure** value of 0.0001 (**Fig.B5**). Click **OK**. Click **OK** again.

Fig.B1

Fig.B2

Fig.B3

Fig.B4

Fig.B5
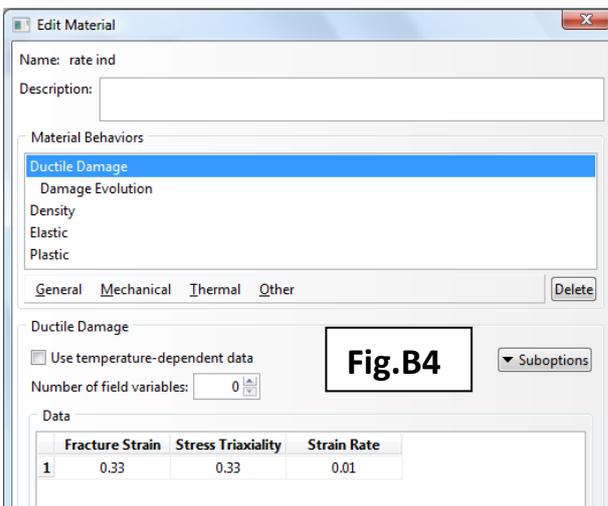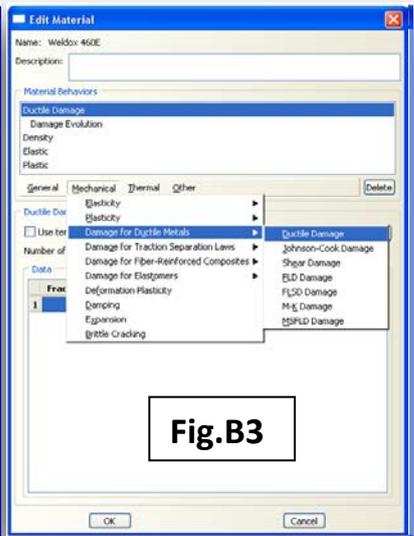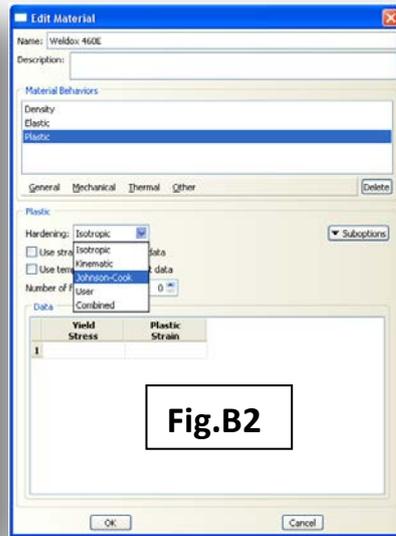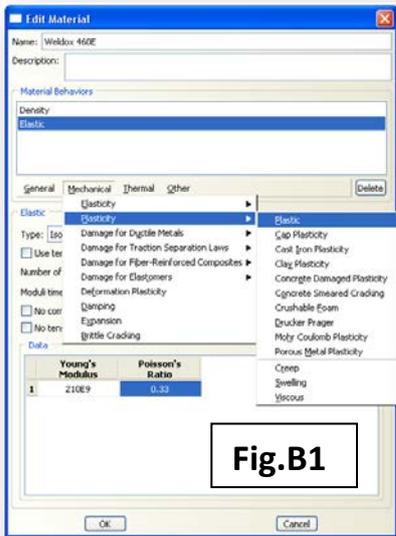
2. From the main menu bar select **Section→Create**. The **Create Section** dialogue box will open. Fill in the **Create Section** dialogue box as shown in **Fig.B6** and click **Continue**.

3. The **Edit Section** dialogue box will appear (**Fig.B7**). You must now define the thickness of the plate (0.4 mm). Click **OK**.

4. From the main menu bar select **Assign→Section**. Using the mouse cursor, click and drag across the plate and click **Done** in the prompt area. Click **OK** in the **Edit Section Assignment** dialogue box. Click **Done** in the prompt area.

Fig.B6

Fig.B7
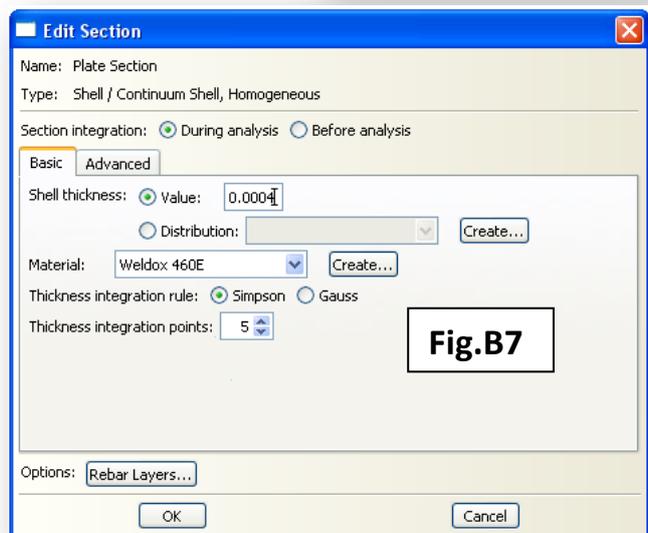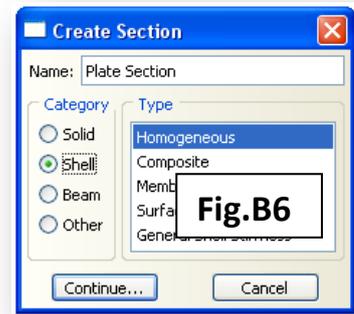
**Whilst in the Property module, you must also specify the projectile mass!!

5. From the main menu bar, select **Special→Inertia→Create**. The **Create Inertia** dialogue box will open. Name the mass condition `Projectile Mass` and select **Point Mass** as the **Type**. Click **Continue**. Select the **Reference Point** you created in the **Part** module as the point to which you wish to assign mass. Click **Done** in the prompt area and the **Edit Inertia** dialogue box will open. Select a mass of 0.002 kg and click **OK**.

C. MODULE→ASSEMBLY

1. From the main menu bar select **Instance→Create**. The **Create Instance** dialogue box will open. Select both of the parts and choose **Independent** as the **Instance Type**. Click **OK**.

2. Translate the projectile in the **Positive Z** direction by 6 mm (**Instance→Translate**). Click **OK** in the prompt area.


D. MODULE→STEP

1. From the main menu bar, select **Step→Create**. The **Create Step** dialogue box will appear Name the step `Impact Step`.

2. Select **General** from the **Procedure Type** options.

3. Select **Dynamic, Explicit** from the list of analysis types. Click **Continue**. The **Edit Step** dialogue box will open. Choose a time period of 0.0002 s (200 µs). Toggle on **NLGEOM**. Click **OK**.

4. Failed elements will need to be removed from the mesh in order to monitor crack propagation in your analyses. In the **Field Output Requests**, toggle on **Status**, which can be found under the **State/Field/User/Time** options.

5. To monitor the projectile velocity, create a new **History Output Request** called `Projectile Velocity`. Choose `Projectile  Set` for the **Domain** and select V3, from the V options underneath the **Displacement/Velocity/Acceleration** list.

E. MODULE→INTERACTION

1. From the main menu bar select **Interaction→Create**. The **Create Interaction** dialogue box will open.

> (a) Name the interaction `Projectile Plate Contact`.
>
> (b) Choose **Surface-to-Surface** for the **Types for Selected Step**.
>
> (c) Click **Continue.**

2. You will be prompted to select the first (**Master**) surface. Using the mouse cursor, select the surface of the analytical rigid projectile and click **Done** in the prompt area.

3. When prompted, choose **Brown** in the prompt area, which represents the outer surface of the projectile.

4. You now need to define the second (**Slave**) surface involved in the contact. Select **Surface** from the prompt area. Using the mouse cursor, select the inner, partitioned region of the plate (**Fig.E1**). Click **Done** in the prompt area. Select **Brown** when prompted to choose a side for the shell surface.

5. The Edit Interaction dialogue box will open (**Fig.E2**). From the **Discretisation Method** select **Surface to Surface**.
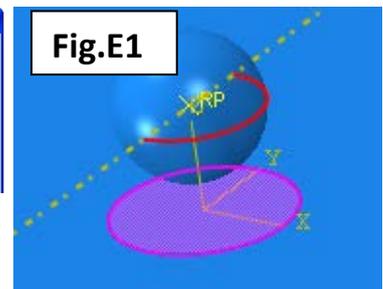
6. Click the **Create** tab next to **Contact Interaction Property**. The **Create Interaction Property** dialogue box will open.

Fig.E1

Fig.E2

7. Name it `Contact Interaction` and select **Contact** as the **Type**. Click **Continue** and the **Edit Contact Property** box will open. Select **Mechanical→Normal**, and accept the default options by clicking **OK**. Click **OK** again.

F. MODULE→LOAD

---

1. From the main menu bar select **BC→Create**. The **Create Boundary Condition** dialogue box will open (**Fig.F1**)

      (a) Name the boundary condition `Clamp`.

      (b) Choose **Mechanical** for the category.

      (c) Choose **Symmetry/Antisymmetry/Encastre** for the **Types for Selected Step**.

      (d) Click **Continue**.

      (e) You must now select the region for the **Clamp** boundary condition as shown in **Fig.F2**.

Fig.F1

Fig.F2
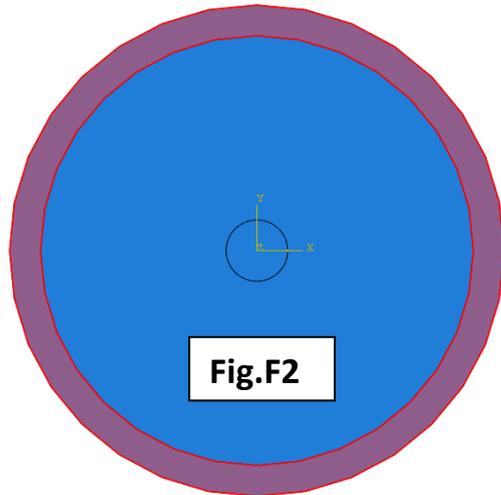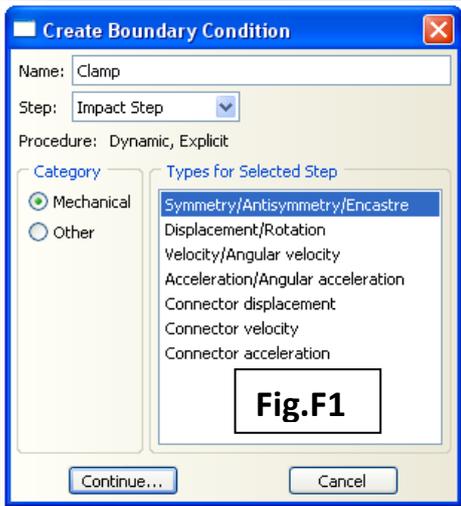
2. Click **Done** in the prompt area. The **Edit Boundary Condition** dialogue box will open. Select **Encastre** and click **OK**.

3. Create a further boundary condition that will allow the projectile to move in the **Z-direction (axis-3)** only. Assign this boundary condition to the projectile **Reference Point** that you created earlier.

4. To define the projectile velocity, from the main menu bar select **Predefined Field→Create**. Select the options shown in **Fig.F3** for the **Create Predefined Field** dialogue box. Click **Continue**.

Fig.F3
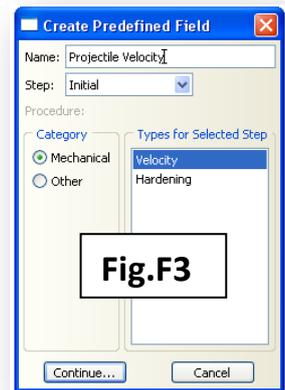
5. When prompted to select the region for the **Predefined Field**, select the projectile **Reference Point** once more. Click **Done** in the prompt area.

6. In the **Edit Predefined Field** dialogue box that appears, choose V1=V2=0 and V3=-150. Click **OK** when done.

G. MODULE→MESH

---

1. From the main menu bar, select **Mesh→Controls**. Using the mouse cursor, click and drag across the plate (remove the projectile using the viewing options!).

     (a) Select **Quad** as the **Element Type**.

     (b) Select **Free** as the **Technique**.

     (c) Select **Medial Axis** as the **Algorithm** type.

     (d) Click **OK**.

2. Using the **Seed** options, seed all 3 edges with 50 seeds using the **Seed Edge by Number** option.

3. From the main menu bar select **Mesh→Element Type**. Click and drag the mouse cursor over the plate and click **Done** in the prompt area. The **Element Type** dialogue box will open. Select **Explicit** from the **Element Library** and **Shell** from the **Element Family** options. Choose **Quad** elements and click **OK** to choose the **S4R** element type. Click **Done** in the prompt area.

4. From the main menu bar select **Mesh→Instance.** Using the mouse cursor, select the plate and click **Done** in the prompt area

**There is no time during the session today to conduct a mesh sensitivity analysis. However, you should bear in mind that, for simulations of this type, the results can be very sensitive to your choice of mesh. Under normal circumstances, you _MUST_ conduct a sensitivity analysis.

G. MODULE→JOB

---

1. Create a job titled `WeldoxBL` and submit the job for analysis. The job should take approximately one to two minutes.

H. MODULE→VISUALISATION

---

1. From the main menu bar select **Results→Field Output**. From the **Field Output** dialogue box select **Mises Stress**. Make a note of whether or not the plate has fully perforated (i.e. **Fig.H1**). If not, increase the impact velocity in the **Load** module. If it has, decrease the impact velocity in the **Load** module. Continue to do this until you converge upon the ballistic limit.

2. The projectile velocity data can be found from the **History Output** options accessible from the main menu bar. Using this information, calculate the absorbed energy (energy lost by the projectile) for impact velocities of 150, 200, 300, 400 500 and 600 m s^{-1}.
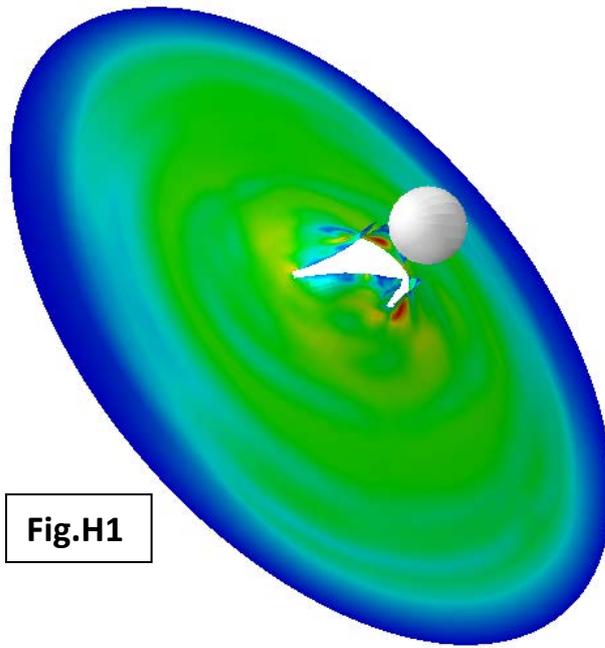
Fig.H1

Solution (b):

- Return to the **Property** module. Using the strain rate hardening and strain rate fracture data from Table I and Table II, define strain rate-hardening behaviour in your material model

---

A. MODULE→PROPERTY

---

1. From the main menu bar select **Material→Edit→Weldox 460E**.

    (a) Select **Mechanical→Plasticity→Plastic**.

    (b) Under the **Suboptions** tab select **Rate Dependent**.

    (c) Under **Hardening** select **Johnson-Cook** and fill in the rate-hardening data from Table I

2. Select **Ductile Damage** from the **Material Behaviours** box. Fill in the rate-dependent fracture strain data as shown in Table II.

B. MODULE→JOB

---

1. Determine the ballistic limit of the plate with these new material property characteristics by submitting the necessary jobs. *You may need to alter the step time for low impact velocities. Submit 5 further jobs at impact speeds between the ballistic limit and 600 m s^{-1}.

C. MODULE→VISUALISATION

---

1. View the **Mises Stress** contours by selecting **Results→Field Output** from the main menu bar. Calculate the absorbed energy by extracting the projectile velocity history.

OPTIONAL QUESTIONS

1. Determine the ballistic limit of the plates when struck at an angle of 45°.

2. The effects of friction have not been considered in these analyses. Using the FE model, and assuming a friction co-efficient between the projectile and the plate of 0.18, determine whether or not there is a contribution to absorbed energy from friction. How much more significant is friction when the angle of incidence is 45° compared to when the plate is struck at normal incidence for an impact velocity of 300 m s^{-1}. **(*Hint; return to your contact interaction property and search under tangential behaviour.)**

3. The strain and strain rate-dependent behaviour has been described using the Johnson-Cook constitutive relation. Is there an alternative way of defining this data in the material editor?

4. In the strain rate-dependent simulations, you defined rate-dependent fracture strains for strain rates as high as 10000 s^{-1}. Is this strain rate-dependent fracture data sufficient to cover the range of strain rates that are generated at the highest impact velocities? **(*Hint; you will need to edit the field output requests to include strain rates as output data.)**