Fully-Coupled Thermo-Mechanical Analysis

Type of solver: ABAQUS CAE/Standard

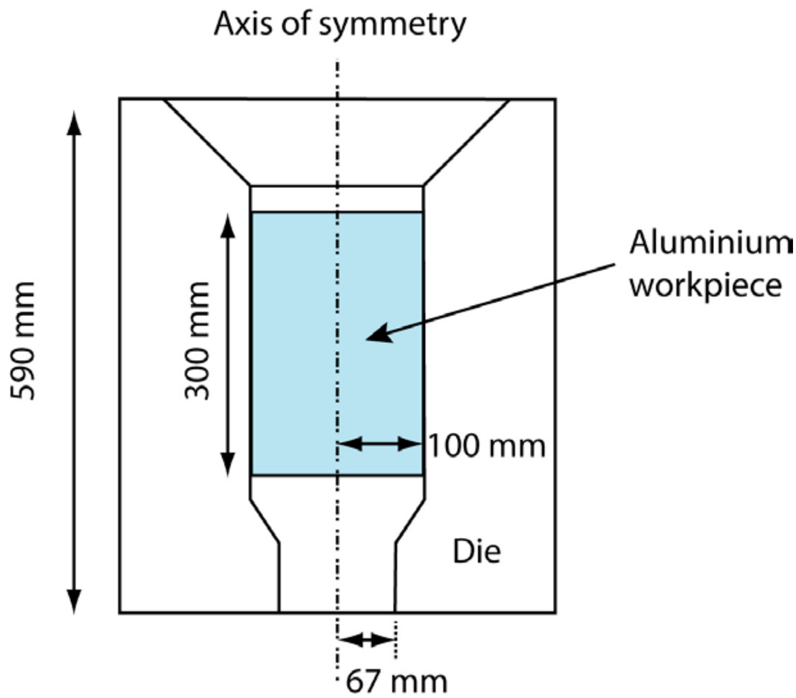
Adapted from: ABAQUS Example Problems Manual

Extrusion of a Cylindrical Aluminium Bar with Frictional Heat Generation

Problem Description:

The figure below shows the cross-sectional view of an aluminium cylindrical bar placed within an extrusion die. The bar has an initial radius of 100 mm and a length of 300 mm, and its radius is to be reduced by 33% through an extrusion process. The die can be assumed as an isothermal rigid body. During the extrusion process, the bar is forced downwards by 250 mm at a constant displacement rate of 25 mm s^{-1}. The generation of heat attributable to plastic dissipation inside the bar and the frictional heat generation at the die-workpiece interface causes temperature of the workpiece to rise. When extrusion is completed, the workpiece is allowed to cool in the ambient air. The ambient surrounding is at 20 °C, with a coefficient of heat transfer of 10 W m^{-2} K^{-1}.

Formulate an axisymmetric FE model to predict (i) the geometry of the deformed bar, (ii) the plastic strain distribution and (iii) the temperature evolution, at various stages of the extrusion process.



SOLUTION:

- Start ABAQUS/CAE. At the **Start Session** dialog box, click **Create Model Database**.
- From the main menu bar, select **Model→Create**. The **Edit Model Attributes** dialog box appears, name the model `TM_Coupled`

A. MODULE → PART

We will construct an axisymmetric model consisting of a deformable workpiece and a rigid die.

(a) To sketch the aluminium alloy workpiece

1. From the main menu bar, select **Part→Create**
2. Name the part `Workpiece`. Use the settings are shown in **Fig.A1**. Ensure that the **Modeling Space** is set to **Axisymmetric** and the **Type** as **Deformable**.
3. Sketch the `Workpiece`, the 4 vertices as shown in **Fig.A2** are (0,0), (0.1,0), (0,0.3) and (0.1,0.3) in metres.

   Note: When building an axisymmetric model, it is important to observe the position of various parts in relation to the axis of symmetry.

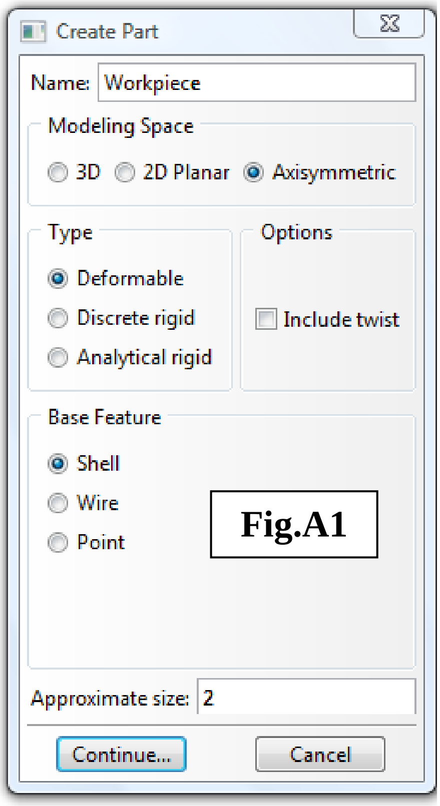


Fig.A1

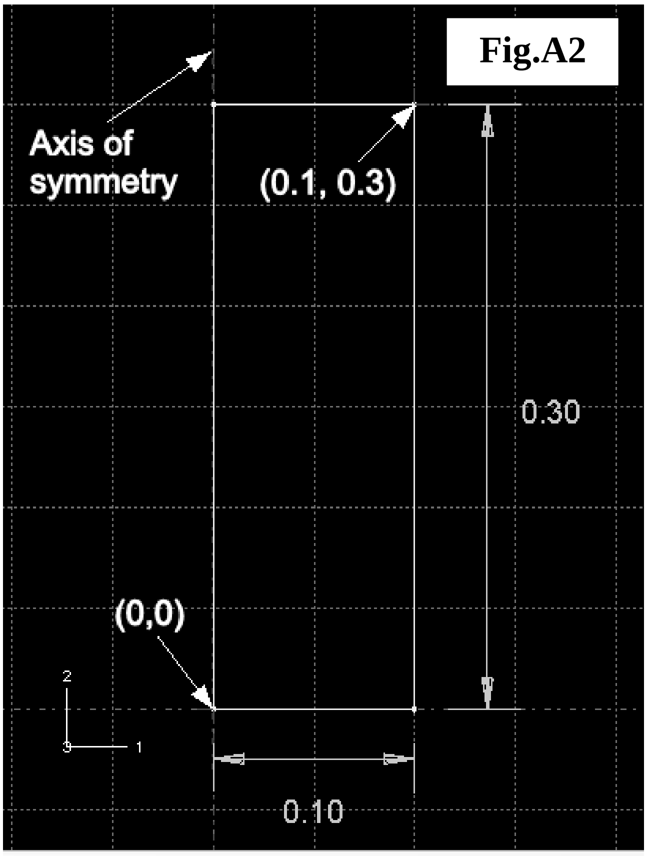


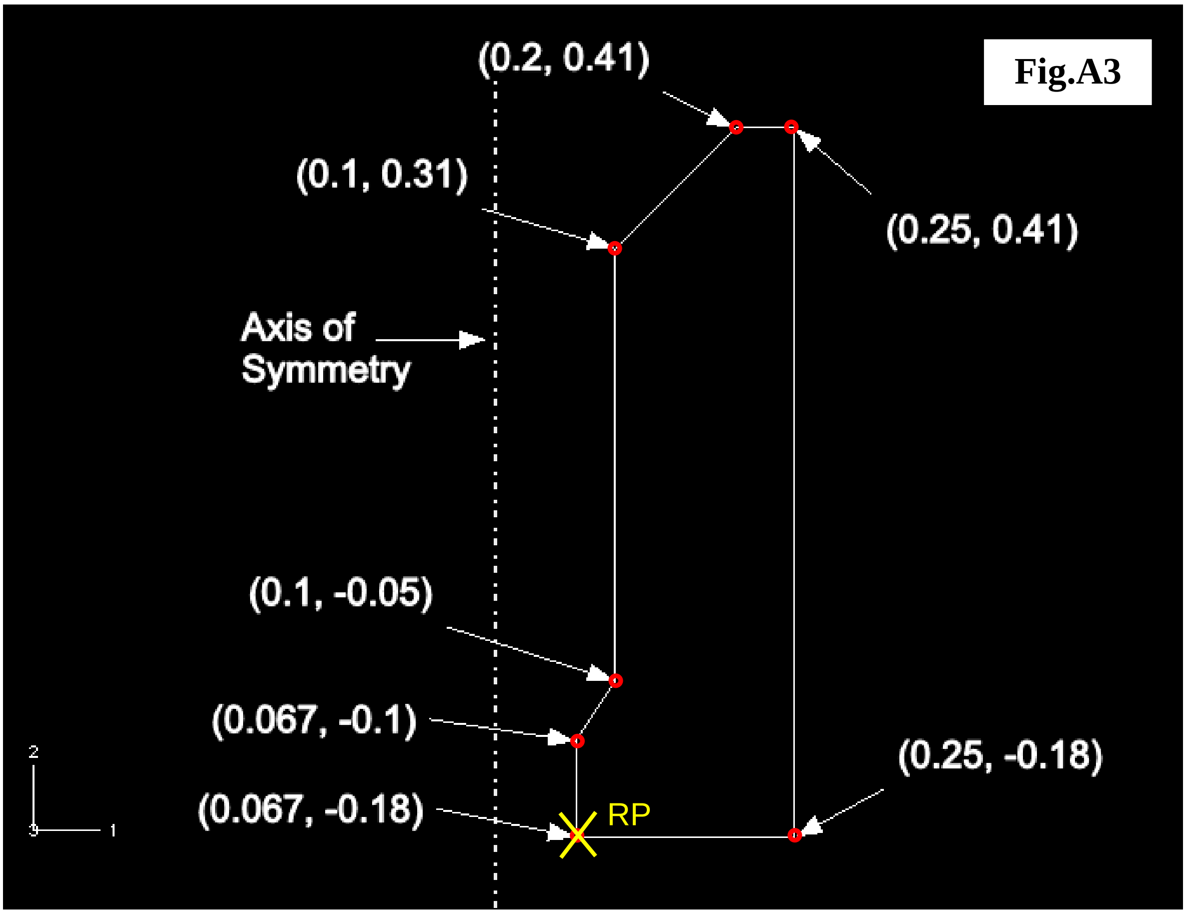
Fig.A2

(b) To sketch the rigid die

1. From the main menu bar, select **Part→Create**
2. Name the part `Die`. Apart from the name, all the other settings are the same as in **Fig.A1**.

   Note: Although the die is meant to be a rigid body in this analysis, here we choose to first build it as a deformable body and later apply a **Rigid body** constraint (Section E (c)).
3. Sketch the `Die` using the vertices given in **Fig.A3**.

   Note: Observe that all coordinates are followed correctly, so that the assembly of the workpiece and die can be carried out correctly later.
4. We also need to add a reference point to the die part to be used in rigid body constraint. From the main menu bar, select **Tools→Reference Point**. Pick point (0.067, -0.18) as denoted in **Fig.A3**, note that a yellow **RP** symbol appears.



B. MODULE → PROPERTY

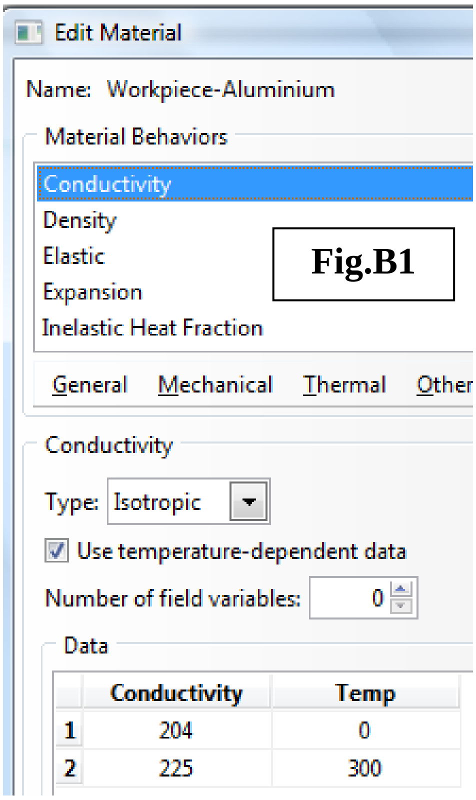
(a) To enter material properties of the workpiece:-

1. From the main menu bar, select **Material→Create**
2. Name the material `Aluminium`.
3. Create the following material properties:

(i) **General→Density** `2700` (kg m^{-3})

(ii) **Thermal→Conductivity**

- Under **Type** choose **Isotropic**
- Toggle on **Use temperature-dependent data** (NB. Conductivity in W m^{-1} K^{-1}, Temp in °C) and use data shown in **Fig.B1**.



(iii) **Thermal→Inelastic Heat Fraction** `0.9`

(iv) **Thermal→Specific Heat** `880` (J kg^{-1} K^{-1})

(v) **Mechanical→Elastic**

Under **Type** choose **Isotropic**

Young's Modulus: 69×10^{9} (Pa)

Poisson's Ratio: `0.33`

(vi) **Mechanical→Expansion**

Under **Type** choose **Isotropic**

Reference temperature: `20` (°C)

Expansion Coeff alpha: 8.42×10^{-5} (K^{-1})

(vii) **Mechanical→Plasticity→Plastic**

Under **Hardening** choose **Isotropic** (**Fig.B2**)

Toggle on **Use Temperature-dependent data**

The complete set of data is given in **Table 1**.

(*Note:* The list of data can also be directly imported into ABAQUS/CAE if an ASCII text file is available (*NOT provided in this exercise*). To do this, right click within the table and choose **Read from File**)

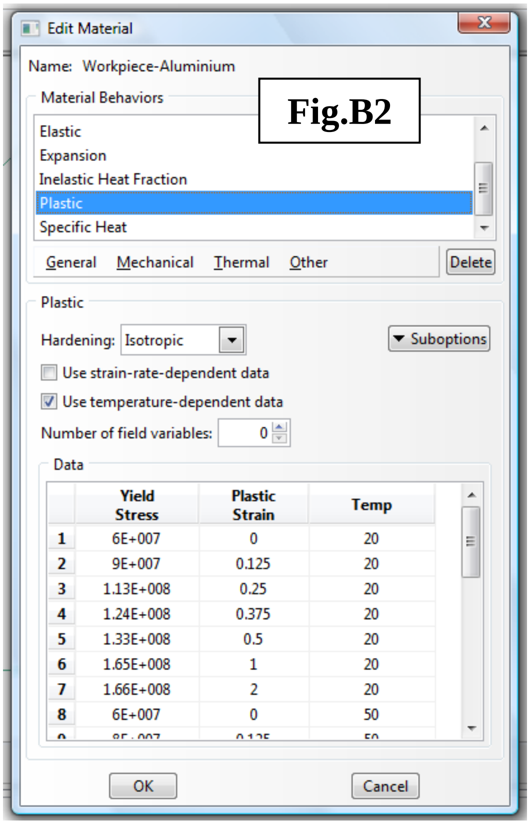


Fig.B2

Table 1: Temperature-dependent flow stress of Aluminium

Yield stress	Plastic strain	Temp
6.00E+07	0	20
9.00E+07	0.125	20
1.13E+08	0.25	20
1.24E+08	0.375	20
1.33E+08	0.5	20
1.65E+08	1	20
1.66E+08	2	20
6.00E+07	0	50
8.00E+07	0.125	50
9.70E+07	0.25	50
1.10E+08	0.375	50
1.20E+08	0.5	50
1.50E+08	1	50
1.51E+08	2	50
5.00E+07	0	100
6.50E+07	0.125	100
8.15E+07	0.25	100
9.10E+07	0.375	100
1.00E+08	0.5	100
1.25E+08	1	100
1.26E+08	2	100
4.50E+07	0	150
6.30E+07	0.125	150
7.50E+07	0.25	150
8.90E+07	0.5	150
1.10E+08	1	150
1.11E+08	2	150

(b) To enter material properties of the die:-

1. From the main menu bar, select **Material→Create**
2. Name the material `Die-Material`
4. Create the following material properties:

   Note: Since the die will be modelled as a rigid body and heat flow into the die is not modelled, the properties entered here will be inconsequential. However, non-zero values must be entered so that the ABAQUS/CAE solver can proceed.

   (i) **General→Density** `2700` (kg m^{-3})

   (ii) **Thermal→Conductivity** `200` (W m^{-1} K^{-1})

   (iii) **Thermal→Specific Heat** `880` (J kg^{-1} K^{-1})

   (iv) **Mechanical→Expansion**

   Under **Type** choose **Isotropic**

   Reference temperature: `20` (°C)

   Expansion Coefficient - alpha: 8.42×10^{-5} (K^{-1})

   (v) **Mechanical→Elastic**

   Under **Type** choose **Isotropic**

   Young's Modulus: 200×10^{9} (Pa)

   Poisson's Ratio: `0.3`

(c) To create the sections and assign them to the parts

1. From the main menu bar, select **Section→Create**
2. Name it `Section-workpiece` (**Fig.B3**). For **Category,** choose **Solid,** and set **Type** as **Homogeneous**.

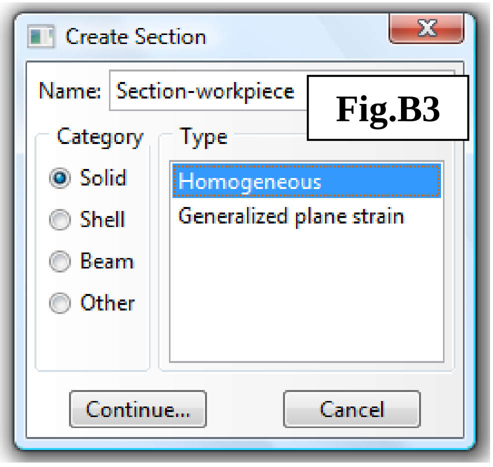


Fig.B3

3. In the **Edit Section** dialogue box (**Fig.B4**), under **Material** pick `Aluminium`.

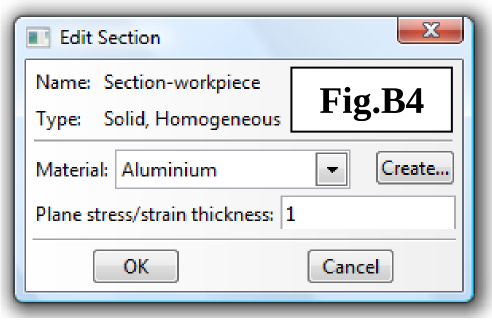


Fig.B4

4. Now create a section for the die, call it `Section-die`.
5. Assign the sections to the relevant parts.

C. MODULE → ASSEMBLY

1. From the main menu bar, select **Instance→Create**
2. First create an instance of the `Die` part. Under **Instance Type**, make sure to select **Independent (mesh on instance)**. Toggle off **Auto-offset from other instances**.
3. Then create an instance of the `Workpiece` part, also as an independent instance. Make sure that **Auto-offset from other instances** is set as off, see **Fig.C1.**
4. The complete assembly of the die and workpiece is depicted on the left panel of **Fig.C1**.

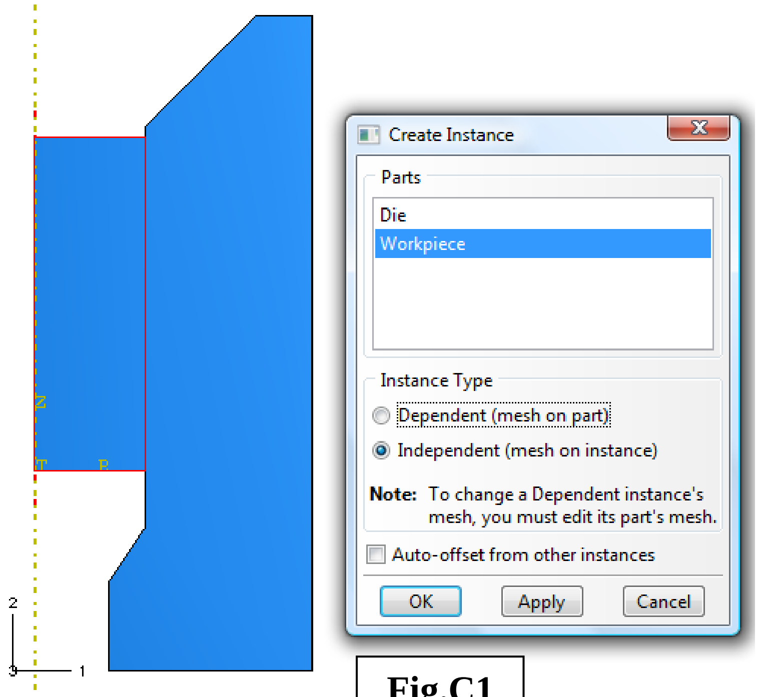


Fig.C1

D. MODULE → STEP

This fully-coupled thermal displacement transient analysis will consist of an initial step (exist by default) plus 4 additional steps (to be created in this section). **Fig.D1** shows the **Step Manager** dialogue box with all the steps correctly set up.

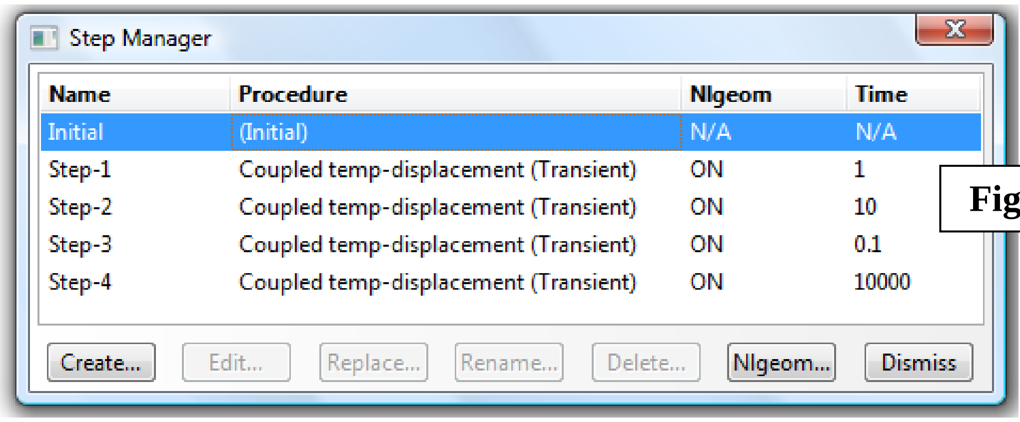
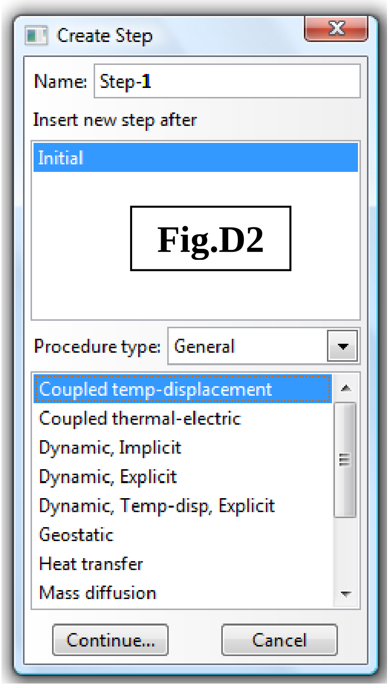


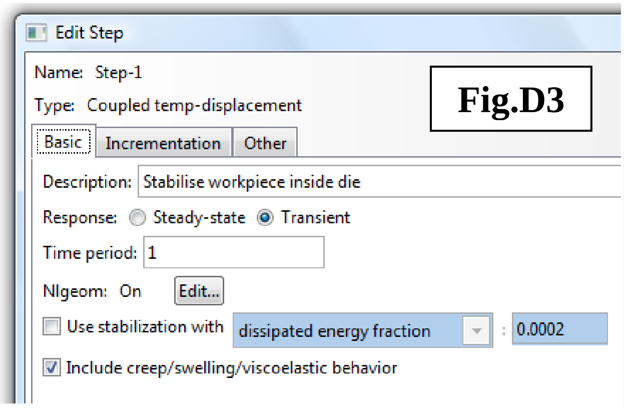
Fig.D1

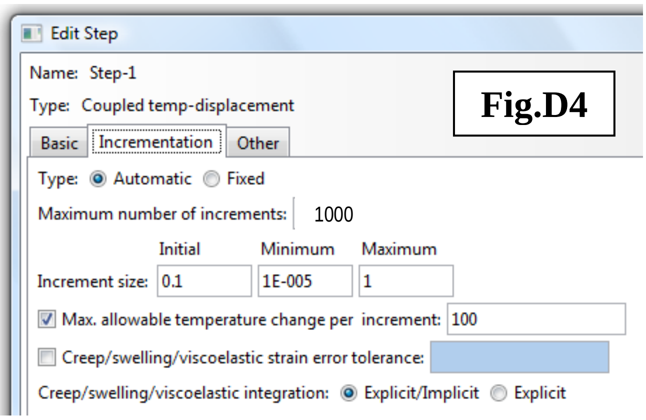
Important: The "**Nlgeom"** option must be enabled to account for large strain plastic deformations.

(a) To create Step-1: Stabilise workpiece inside die



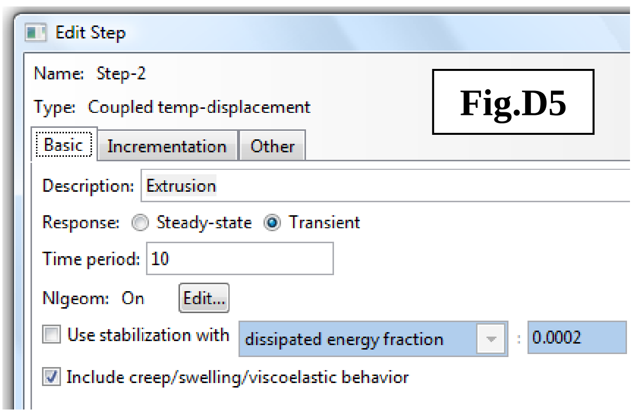
1. From the main menu bar, select **Step→Create**
2. Name it `Step-1` (**Fig.D2**). The **Procedure type** is **General→Coupled temp-displacement**
3. In **Edit Step** dialog box, under the **Basic** tab (**Fig.D3**), enter `Stabilise workpiece inside die` as the **Description**. To account for large plastic deformation, toggle on **Nlgeom**. To consider time-dependent plasticity, toggle on **Include creep/swelling/viscoelastic behavior**.
4. In **Edit Step** dialog box, click on the **Incrementation** tab (**Fig.D4**) and reduce the **Initial Increment size** to `0.1`. Toggle on **Max. allowable temperature change per increment** and enter `100`.
5. Accept the default settings under the **Other** tab.

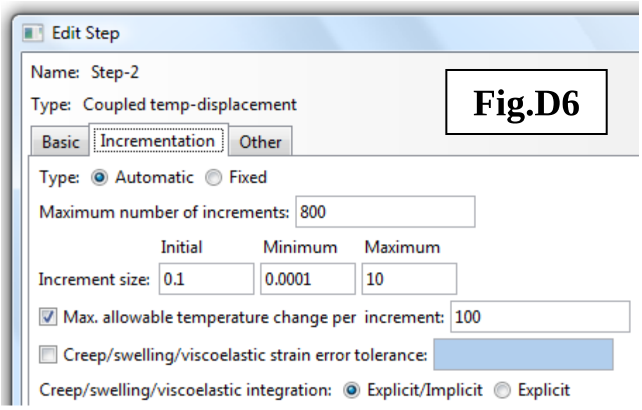




(b) To create Step-2: Extrusion

Create `Step-2`. As of `Step-1`, the **Procedure type** is **Coupled temp-displacement**. **Figs.D5** and **D6** show the parameters to be used.





(c) To create Step-3: Remove contact pairs

Create `Step-3` and fill out the parameters as in **Figs.D7** and **D8**.

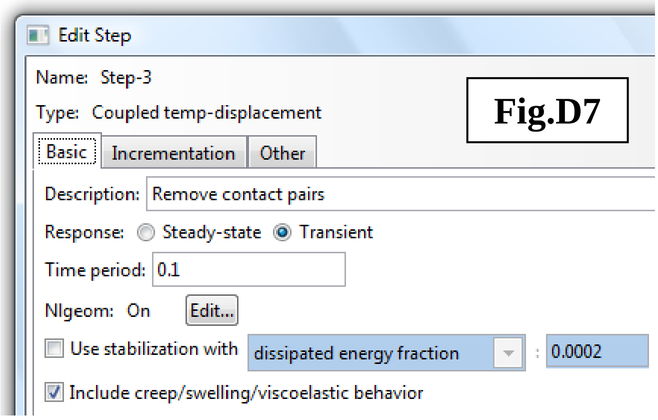


Fig.D7

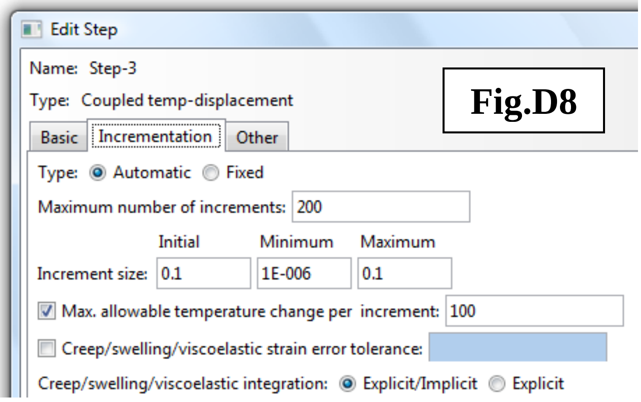


Fig.D8

(d) To create Step-4: Let workpiece cool down

Create `Step-4` and fill out the parameters as in **Figs.D9** and **D10**.

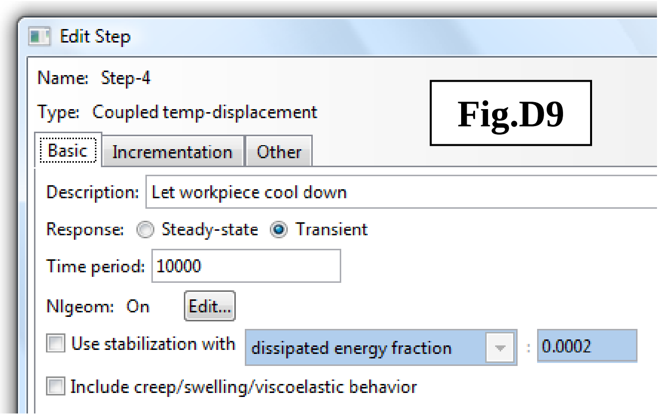


Fig.D9

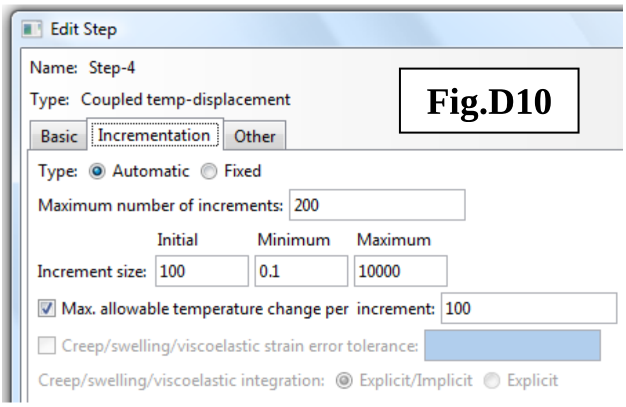


Fig.D10

E. MODULE → INTERACTION

(a) To create surfaces for interaction

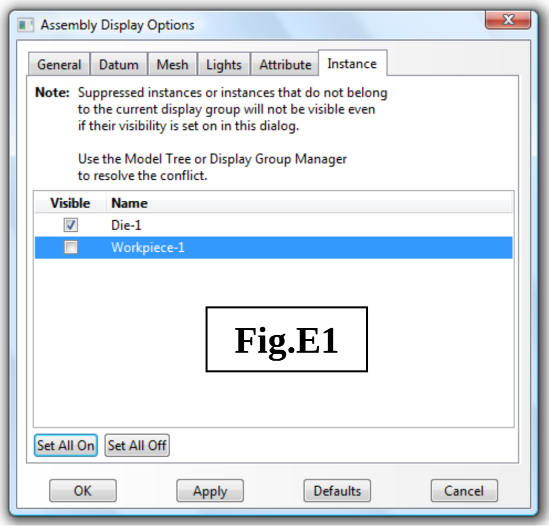


Fig.E1

1. From the main menu bar, select **View→Assembly Display Options**
2. Click the **Instance** tab, toggle off the visibility of `Workpiece-1`, see **Fig.E1**. Click **OK**.
3. From the main menu bar, select **Tools→Surface→Create.**
4. Name the surface `Surf-die-Contact`, pick the 5 edges designated in **Fig.E2**. *Tip:* To make multiple selections, hold down the Ctrl-button while clicking.
5. Return to **Assembly Display Options** to toggle on the visibility of `Workpiece-1`, then toggle off `Die-1`. Click **OK**.
6. Create a surface called `Surf-workpiece-Vertical`, i.e. the vertical edge that comes into contact with the die, see **Fig.E3**.
7. Create a surface called `Surf-workpiece-Horizontal`, as designated in **Fig.E3**.
8. Finally, create another surface called `Surf-workpiece-Convect` that consists of 3 edges denoted in **Fig.E4**.
9. Toggle on both instances when finished assigning all surfaces.

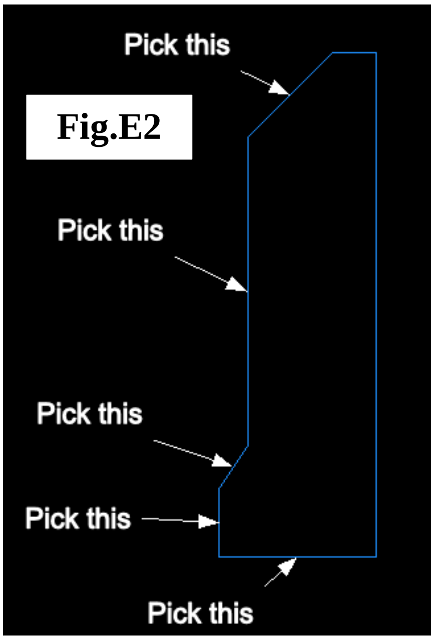


Fig.E2

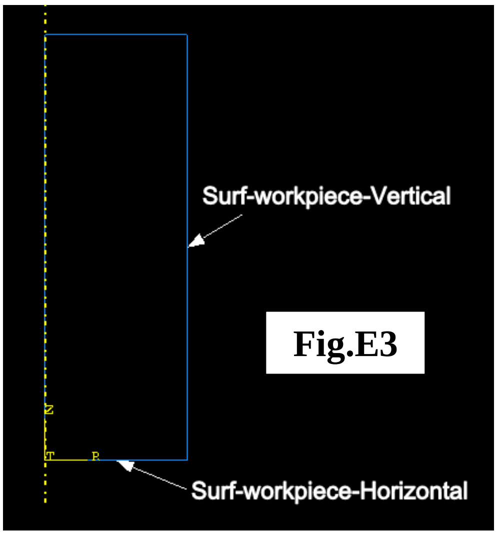


Fig.E3

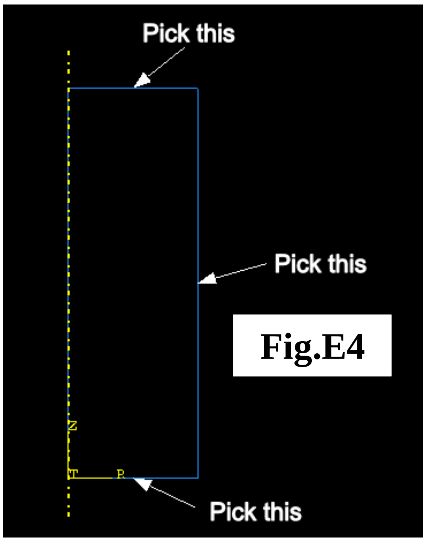


Fig.E4

(b) To create the interaction property

1. From the main menu bar, select **Interaction→Property→Create**
2. Name it `IntProp-1`. Under **Type**, select **Contact**.
3. In the **Edit Contact Property** dialogue box (**Fig.E5**), add the following properties:

   (i) **Mechanical→Tangential Behavior**

   - For **Friction formulation**, choose **Penalty**
   - **Friction Coeff**: `0.1`

   (ii) **Thermal→Heat Generation**

   - Use `0.5`, `0.5`

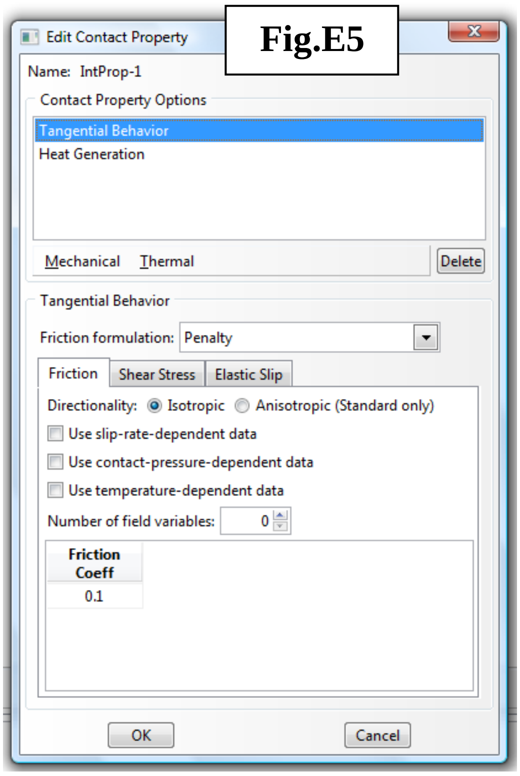


Fig.E5

(c) To create a rigid body constraint for the die

1. From the main menu bar, select **Constraint →Create**
2. Name the constraint `Die-RigidBody`. Under **Type**, select **Rigid body**.
3. The **Edit Constraint** dialogue box appears (**Fig.E6**). Under **Region type**, choose **Body(elements)** and then pick the die region.
4. For **Reference Point**, pick the yellow point denoted as **RP** (see **Fig.A3**), i.e. the reference point of the die.
5. Toggle on **Constraint selected regions to be isothermal**.

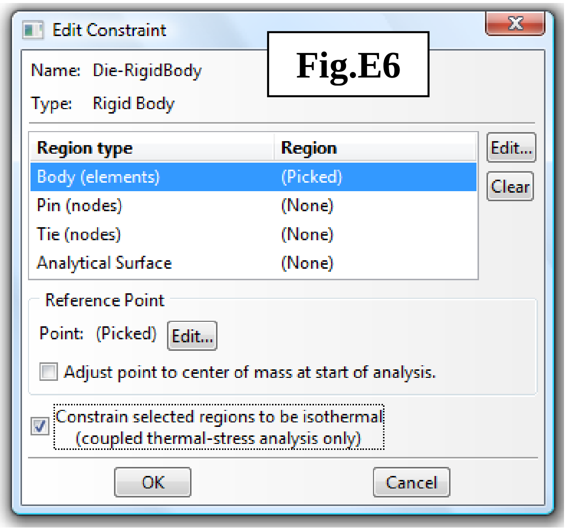


Fig.E6

(d) To create the interactions

We will create 3 interactions, the end result is shown in **Fig.E7**. Note that not all interactions are active at all steps.

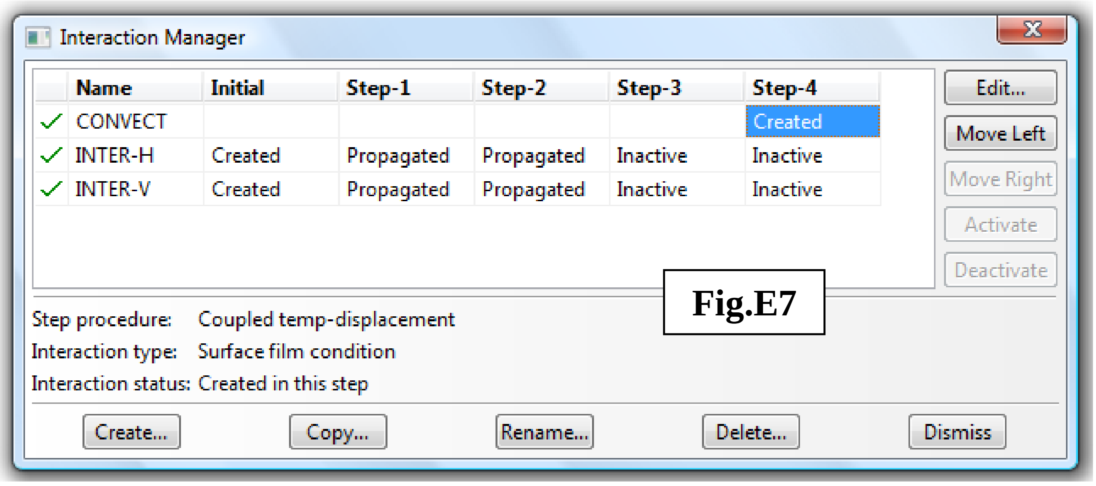


Fig.E7

- ***Thermal film interaction***

1. From the main menu bar, select **Interaction →Create**
2. Name it `CONVECT`. Select `Step-4` and **Surface film condition**, see **Fig.E8**.
3. To pick the surface, click on the [Surfaces...] button located at the right hand corner of the prompt area, then select `Surface-workpiece-Convect`.
4. In the **Edit Interaction** dialogue box, enter **Film coefficient** as `10` (W m^{-2} K^{-1}) and **Sink temperature** as `20` (°C).

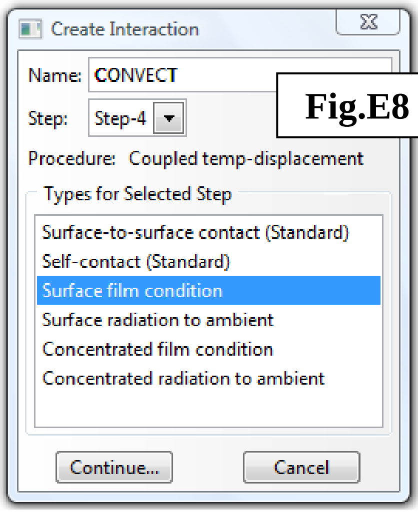


Fig.E8

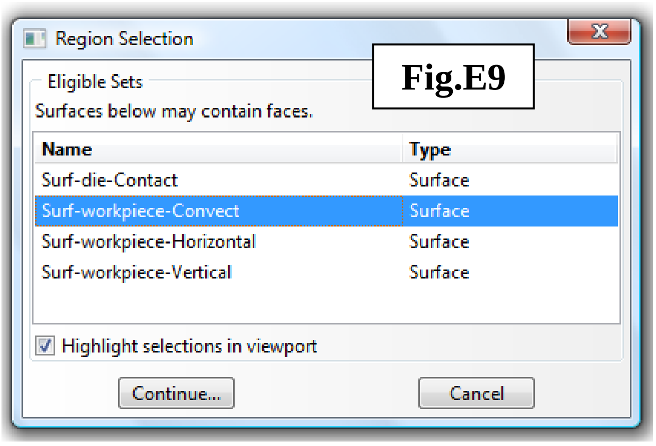


Fig.E9

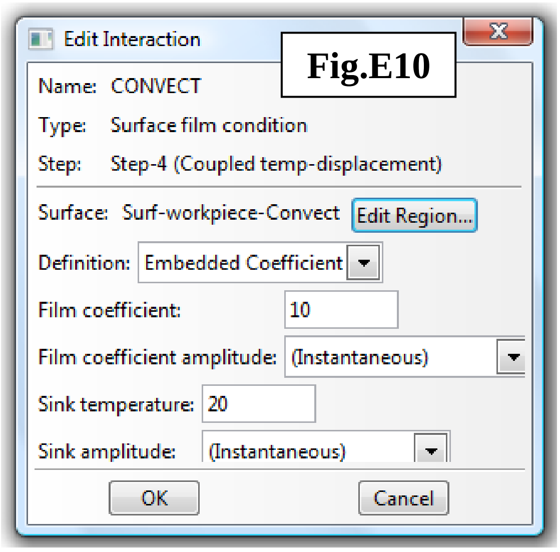


Fig.E10

- ***Mechanical interactions***

1. From the main menu bar, select **Interaction →Create**
2. Name it `INTER-H`. Select **Step:** `Initial` and **Surface-to-surface contact (Standard)**, see **Fig.E11**.
3. For the master surface, select `Surf-die-Contact` (i.e. choose the stiffer of the pair)
4. Choose the slave type as **Surface**, then select `Surf-workpiece-Horizontal`.

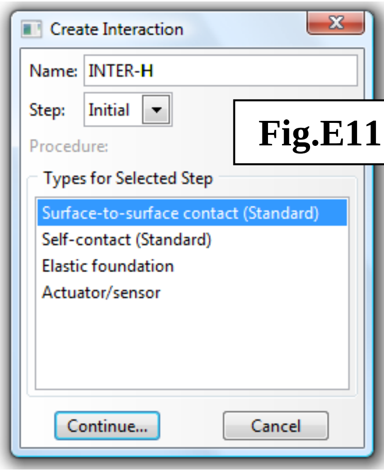


Fig.E11

5. The **Edit Interaction** dialogue box (**Fig.E12**) appears. Set **Degree of smoothing for master surface** as `0.48`. Accept the rest of the default settings. Note that the **Contact interaction property** is `IntProp-1` which was created earlier in *(a)*.

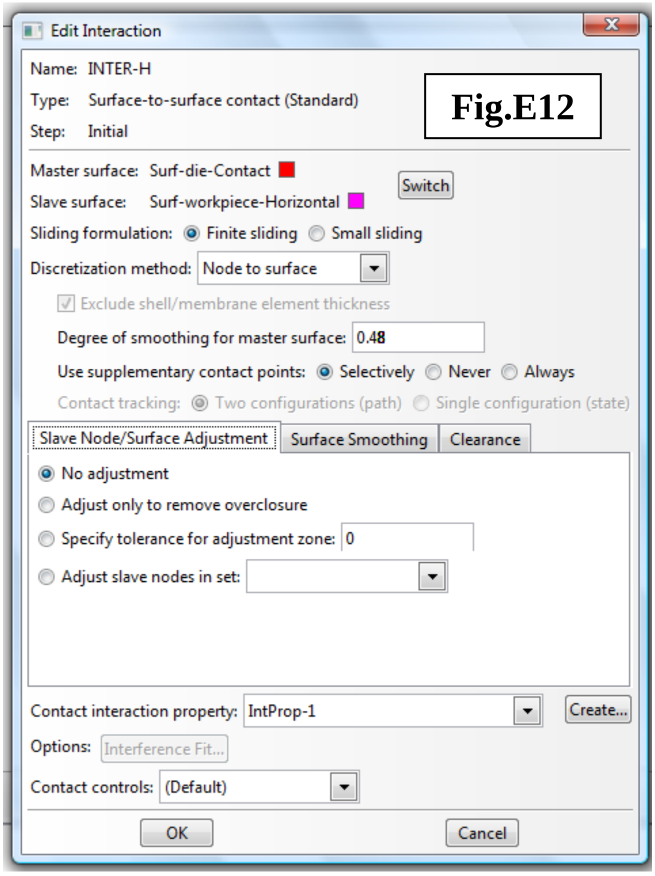


Fig.E12

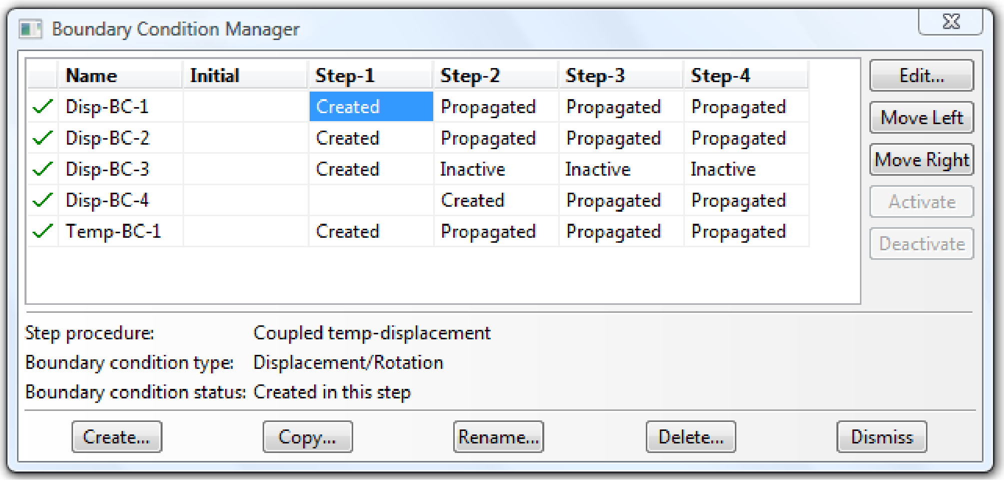
6. Now using similar procedures, create an interaction for `INTER-V`. Assign `Surf-die-Contact` as the master surface and `Surf-workpiece-Vertical` as the slave.
7. We also need to make `INTER-H` and `INTER-V` inactive during `Step-3` (*Remove contact pairs*) and `Step-4` (*Let workpiece cool down*). From the main menu bar, select **Interaction →Manager** to bring up the **Interaction Manager** dialogue box (**Fig.E7**). Click on the box that corresponds to `INTER-H` and `Step-3`, then click on **Deactivate**. Repeat for `INTER-V`.

F. MODULE → LOAD

(a) To create the boundary conditions

- We will create 5 boundary conditions (BCs), the end result is shown in **Fig.F1**. Four of which are displacement boundary conditions (`Disp-BC`) and the last is a temperature boundary condition (`Temp-BC`). Note that not all BCs are active at all times.

Fig.F1



(i) `Disp-BC-1`

1. From the main menu bar, select **BC→Create**
2. Name it `Disp-BC-1` (**Fig.F2**). Assign to **Step:** `Step-1`. **Category**: **Mechanical→Displacement/Rotation**
3. Pick **RP**, the reference point of the die.
4. Set **U1** = **U2** = **UR3** = `0` (To ensure that the die remains static throughout the simulation), see **Fig.F3**.

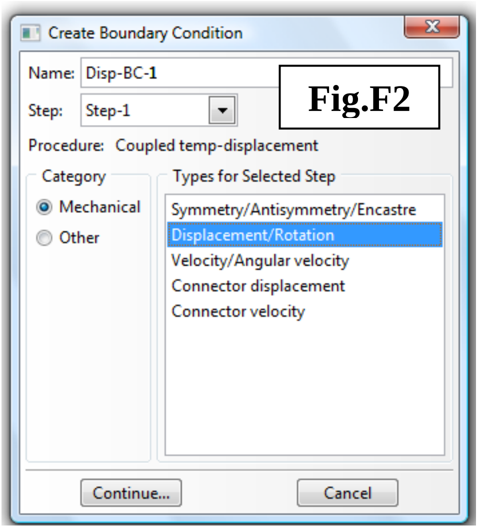


Fig.F2

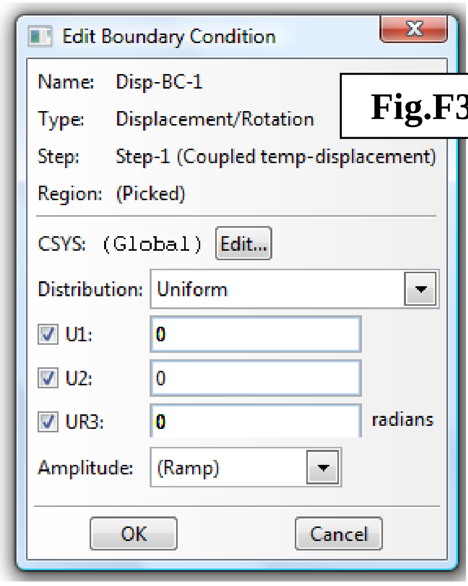


Fig.F3

(ii) `Disp-BC-2`

1. From the main menu bar, select **BC→Create**
2. Name it `Disp-BC-2`. Assign to **Step:** `Step-1`. **Category**: **Mechanical→Displacement/Rotation**
3. Pick the edge corresponding to the axis of the workpiece, see **Fig.F4**.
4. Set **U1** = `0` (To ensure that the workpiece remains axisymmetric throughout the simulation).

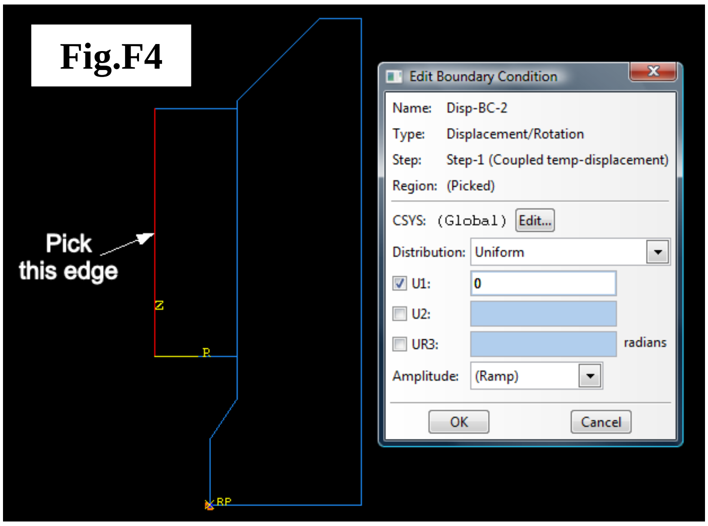


Fig.F4

(iii) `Disp-BC-3`

1. From the main menu bar, select **BC→Create**
2. Name it `Disp-BC-3`. Assign to **Step:** `Step-1`. **Category**: **Mechanical→Displacement/Rotation**
3. Pick the edge corresponding to the top surface of the workpiece, see **Fig.F5**.
4. Set **U2** = `-0.000125` (m)
   Note: A relatively *small* vertical displacement is assigned to the top surface of the workpiece at the start of the simulation to establish contacts at the interfaces.

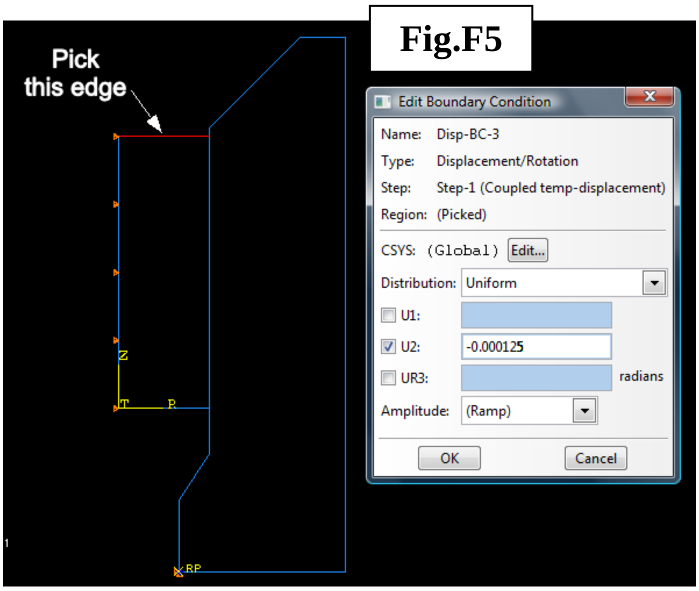


Fig.F5

5. Deactive this BC for `Step-2` and beyond, use the **Boundary Condition Manager** to do this - see **Fig.F1**.

(iv) Disp-BC-4

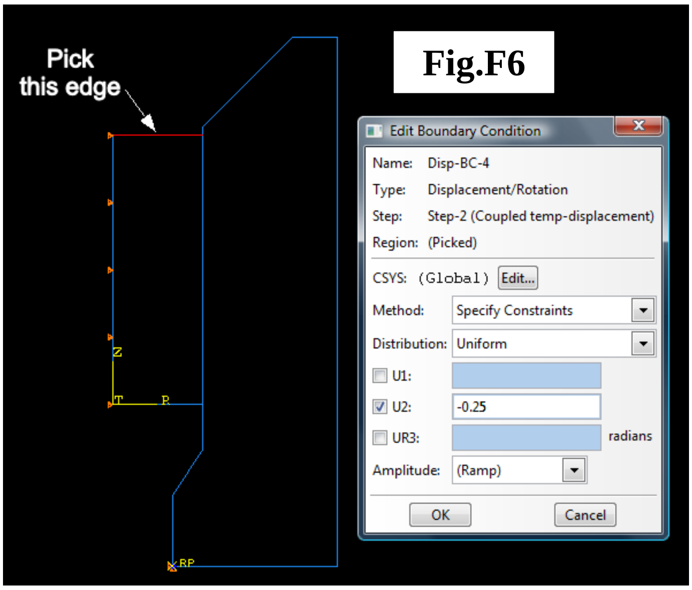


Fig.F6

1. From the main menu bar, select **BC→Create**
2. Name it Disp-BC-4. Assign to **Step:** Step-2. **Category**: **Mechanical→Displacement/Rotation**
3. Pick the edge corresponding to the top surface of the workpiece, see **Fig.F6**.
4. Set **U2** = -0.25 (Displace the workpiece by 250 mm downwards, i.e. to simulate the extrusion process)

(v) Temp-BC-1

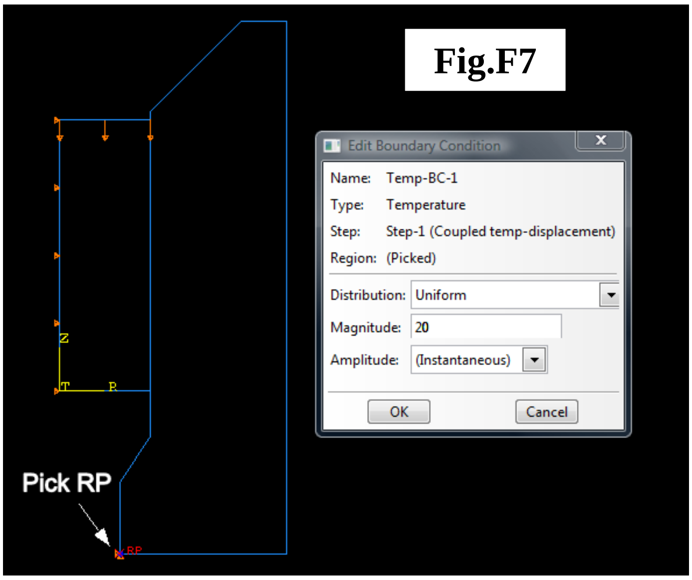


Fig.F7

1. From the main menu bar, select **BC→Create**
2. Name it Temp-BC-1. Assign to **Step:** Step-1. **Category: Other→Temperature**
3. Pick the RP on the die, see **Fig.F7**.
4. Set the Magnitude as 20 (°C). The temperature of the die remains constant throughout simulation since we are not accounting for heat transfer into the die.

(b) To create a thermal field for the workpiece

1. From the main menu bar, select **Predefined Field→Create**
2. Name it Field-workpiece. Assign to **Step:** Initial. **Category: Other→Temperature**
3. Pick the Workpiece instance. In the **Edit Field** dialogue box, enter 20 (°C) as the **Magnitude**. This is the initial temperature, the temperature in subsequent steps will be computed.

G. MODULE → MESH

(a) To mesh the workpiece:-

1. First hide the die in the viewport: From the main menu bar, select **Interaction→Assembly Display Options→Instance**, toggle off the die instance.

2. From the main menu bar, select **Seed→Edge By Number**

3. Assign 10 seeds to the horizontal edge and 30 seeds to the vertical edge, see **Fig.G1**.

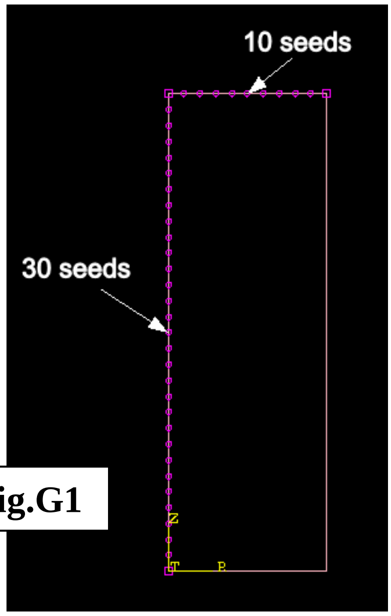


Fig.G1

4. From the main menu bar, select **Mesh→Controls**. Use **Quad** elements and apply **Structured** (green) technique (**Fig.G2**).

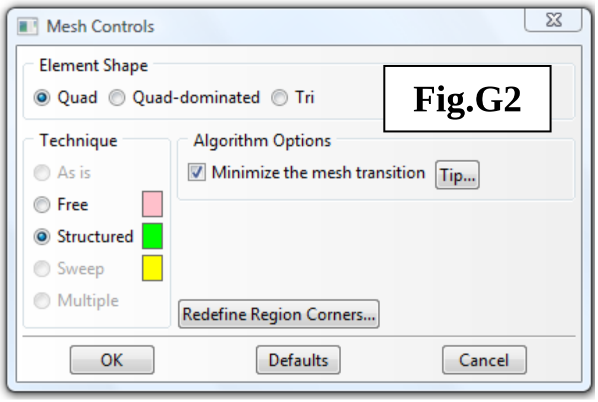


Fig.G2

5. From the main menu bar, select **Mesh→Element Type**. **Element Library**: **Standard**. Under **Family**, choose **Coupled Temperature-Displacement**. Use element type **CAX4T**, see **Fig.G3**.

6. From the main menu bar, select **Mesh→Mesh Instance**. Select the workpiece instance to generate the mesh, it should resemble **Fig.G4**.

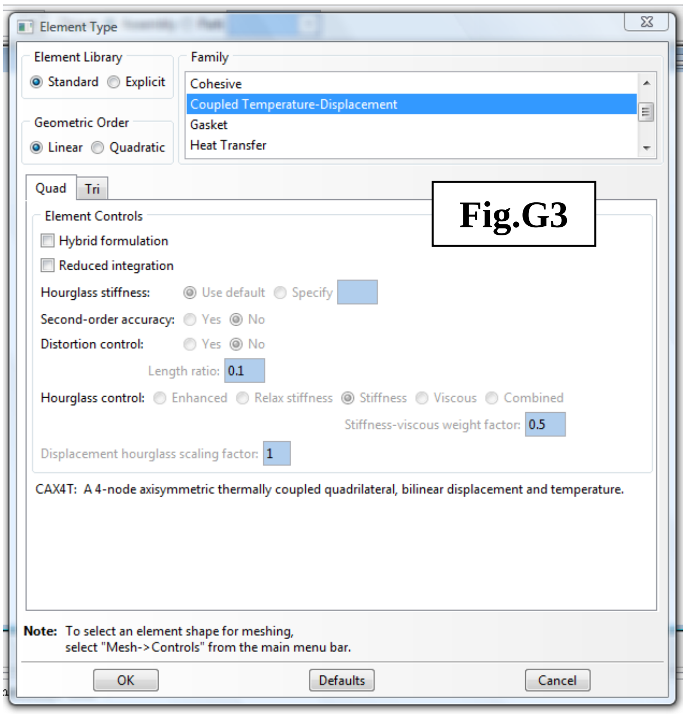


Fig.G3

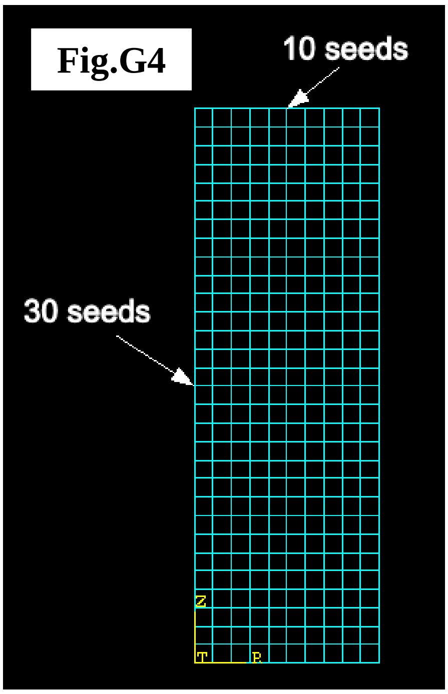


Fig.G4

(b) To mesh the die:-

1. Now hide the workpiece and make the die instance visible.
2. Seed the edges of the die as shown in **Fig.G5**.

   Note: To reduce computation time, we apply relatively coarse mesh for the die since it acts as a rigid body here and heat transfer across the interface is not accounted for in this analysis.
3. Under **Mesh Control**, choose **Quad**, **Structured** and toggle on **Minimize the mesh transition** (**Fig.G2**).
4. Apply **CAX4T** element type.
5. The generated mesh of the assembly is shown in **Fig.G6**.

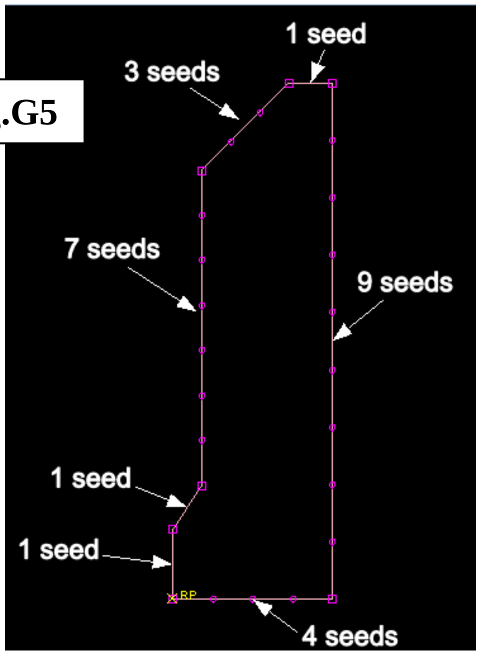


Fig.G5

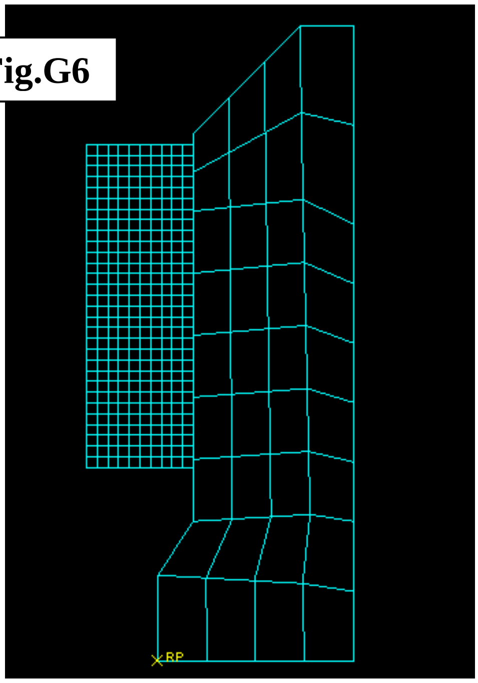


Fig.G6

H. MODULE → JOB

1. From the main menu bar, select **Job→Create**
2. Enter `Job-Extrusion` as the job name.
3. Submit the job and monitor the progress. This analysis can take between 15 and 30 minutes depending on your system.
4. When the job is completed, from the **Job Manager** dialogue box, click on **Results**.

I. MODULE → VISUALIZATION

1. To display the deformed configuration after `Step-2` of the analysis (**Fig.I1**): from the main menu bar, select **Plot→Deformed Shape**, then use the control buttons  in the context bar to scroll through the ODB frames.

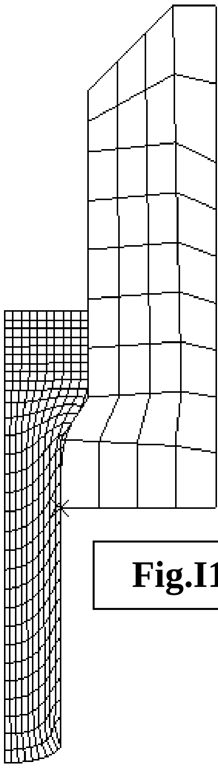


Fig.I1

2. To display plastic strain contours at the end of `Step-2`, select **Result→Field Output**, select **PEEQ**, and use the **Step/Frame** button at the top of the dialogue box to choose the step name (**Fig.I2**).

3. To show the temperature field at the end of `Step-2`, select **NT11** in **Result→Field Output**, the result is shown in **Fig.I3**.

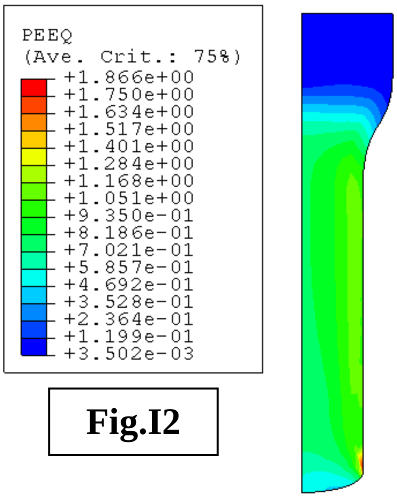


Fig.I2

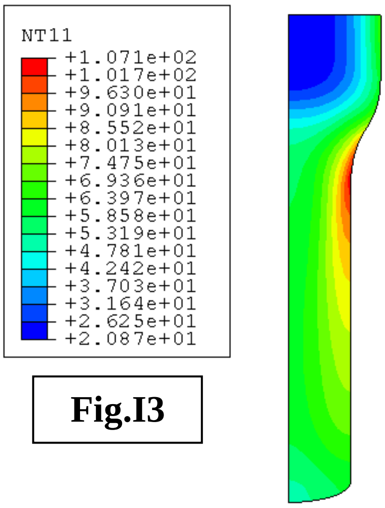


Fig.I3

4. To generate a 3-D view of the axisymmetric model: from the main menu bar, select **View→ODB Display Option**, then click on the **Sweep & Extrude** tab, toggle on **Sweep elements**, enter the angles and segment specification. An example is depicted in **Fig.I4** with angles from 0° to 270°.

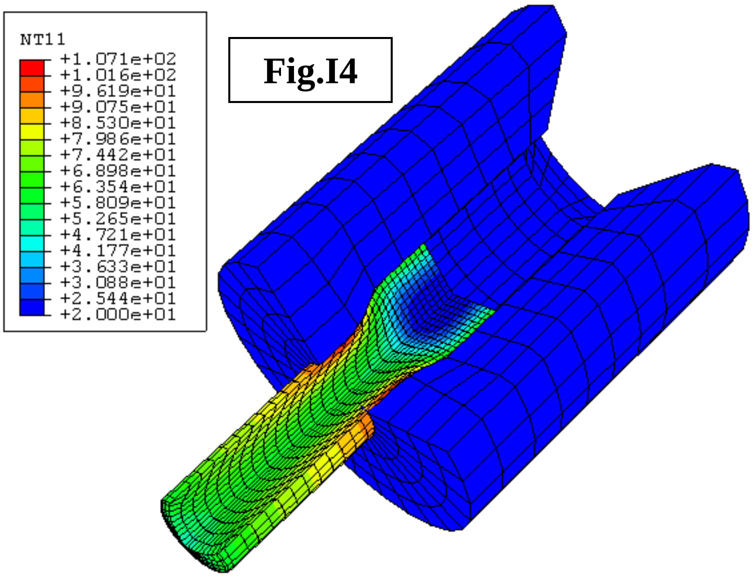


Fig.I4

Optional questions

1. Explore the sensitivity of the model predictions towards the choice of element types and meshing strategies.
2. Show how you can monitor and record the temperature history at a specific node located in the vicinity of the hot zone where the maximum temperature is over 100 °C (see **Fig.I3**).
3. Show how you can model the effects of heat transfer from the workpiece into the die.
4. If you're interested in studying the effects of strain rates, what extra information will be needed?
5. Investigate the contribution of heat generation due to friction towards the overall temperature rise.