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 \rightarrow 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** \rightarrow **Create**
- 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.



Create Part	23
Name: Workpiece	
Modeling Space	
🔘 3D 🔘 2D Plan	ar 💿 Axisymmetric
_ Туре	Options
Oeformable	
Discrete rigid	Include twist
Analytical rigid	1
Base Feature	
Shell	
Wire	
Point	rig.A1
Approximate size:	2
Continue	Cancel

(b) To sketch the rigid die

- 1. From the main menu bar, select **Part** \rightarrow **Create**
- 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)).
- 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.
- 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 \rightarrow PROPERTY

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

- 1. From the main menu bar, select Material \rightarrow Create
- 2. Name the material Aluminium.
- 3. Create the following material properties:
 - (i) General \rightarrow 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** \rightarrow **Specific Heat** 880 (J kg⁻¹ K⁻¹)

(v) Mechanical \rightarrow Elastic

Under **Type** choose **Isotropic Young's Modulus:** 69×10⁹ (Pa) **Poisson's Ratio:** 0.33

(vi) Mechanical \rightarrow Expansion

Under **Type** choose **Isotropic Reference temperature:** 20 (°C) **Expansion Coeff alpha:** 8.42×10⁻⁵ (K⁻¹)

Edit Material	
Name: Workpiece-Alun	ninium
Material Behaviors	
Conductivity	
Density	
Elastic	Fig.B1
Expansion	8
Inelastic Heat Fraction	
<u>G</u> eneral <u>M</u> echanica	l <u>T</u> hermal <u>O</u> ther
Conductivity	
Type: Isotropic 💌	
🔽 Use temperature-de	ependent data
Number of field variab	les: 0 🛓
Data	
Conductivity	Тетр
1 204	0
2 225	300

(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**

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

📧 Edit N	Material				X
Name:	Workpiece-Alumi	inium			
- Materi	al Behaviors		F	ia R2	
Elastic			T.	Ig.D2	•
Expans	ion				
Inelasti	c Heat Fraction				=
Plastic					
Specifi	c Heat				
Gener	al <u>M</u> echanical	<u>T</u> herma	l <u>O</u> th	er	Delete
Harder Use Use Numb Data	ning: Isotropic strain-rate-depe temperature-dep er of field variable	ndent dat pendent d es:	a ata) 🚔		Suboptions
	Yield Stress	Plast Stra	ic in	Temp	^
1	6E+007	0		20	=
2	9E+007	0.12	5	20	
3	1.13E+008	0.2	i	20	
4	1.24E+008	0.37	5	20	
5	1.33E+008	0.5		20	
6	1.65E+008	1		20	
7	1.66E+008	2		20	
8	6E+007	0		50	_
	95.007	0.10	c	50	
	ОК			Cance	4

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

- 1. From the main menu bar, select Material \rightarrow 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 \rightarrow Density 2700 (kg m⁻³)
 - (ii) **Thermal** \rightarrow **Conductivity** 200 (W m⁻¹ K⁻¹)
 - (iii) **Thermal** \rightarrow **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 \rightarrow Elastic

Under Type choose Isotropic

Young's Modulus: 200×10⁹ (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 \rightarrow Create
- Name it Section-workpiece (Fig.B3). For
 Category, choose Solid, and set Type as Homogeneous.
- In the Edit Section dialogue box (Fig.B4), under Material pick Aluminium.
- 4. Now create a section for the die, call it Section-die.
- 5. Assign the sections to the relevant parts.



C. MODULE \rightarrow ASSEMBLY

- 1. From the main menu bar, select **Instance** \rightarrow **Create**
- First create an instance of the Die part. Under Instance Type, make sure to select Independent (mesh on instance). Toggle off Autooffset from other instances.
- 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.
- The complete assembly of the die and workpiece is depicted on the left panel of Fig.C1.



D. MODULE \rightarrow 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.

Step Manage	r			~
Name	Procedure	Nlgeom	Time	
Initial	(Initial)	N/A	N/A	
Step-1	Coupled temp-displacement (Transient)	ON	1	
Step-2	Coupled temp-displacement (Transient)	ON	10	Fig.D
Step-3	Coupled temp-displacement (Transient)	ON	0.1	
· ·	Counted to an effect of the Counter (Townsignat)	ON	10000	

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** \rightarrow **Create**
- Name it Step-1 (Fig.D2). The Procedure type is
 General→Coupled temp-displacement
- 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.
- X 💽 Create Step Name: Step-1 Insert new step after Fig.D2 Procedure type: General • Coupled temp-displaceme * Coupled thermal-electric Dynamic, Implicit Ξ Dynamic, Explicit Dynamic, Temp-disp, Explicit Geostatic Heat transfer Mass diffusion Continue... Cancel

5. Accept the default settings under the **Other** tab.

Edit Step	Edit Step
Name: Step-1 Type: Coupled temp-displacement Basic Incrementation Other Description: Stabilise workpiece inside die Response: Steady-state Increment Time period: 1 NIgeom: On Edit Use stabilization with dissipated energy fraction Increment Include creep/swelling/viscoelastic behavior	Name: Step-1 Type: Coupled temp-displacement Basic Incrementation Other Type: Automatic Fixed Maximum number of increments: 1000 Initial Minimum Maximum Increment size: 0.1 1E-005 1 Image: Max. allowable temperature change per increment: 100 100 Creep/swelling/viscoelastic strain error tolerance: Creep/swelling/viscoelastic integration: Image: Explicit

(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.

Edit Step	Edit Step
Name: Step-2 Type: Coupled temp-displacement Basic Incrementation Other Description: Extrusion	Name: Step-2 Type: Coupled temp-displacement Basic Incrementation Other Type: Automatic Fixed
Response: Steady-state Transient Time period: 10 NIgeom: On Edit Use stabilization with dissipated energy fraction : 0.0002 Image: Include creep/swelling/viscoelastic behavior : 0.0002	Maximum number of increments: 800 Initial Minimum Maximum Maximum Increment size: 0.1 0.0001 10 Image: Max. allowable temperature change per increment: 100 100 Image: Creep/swelling/viscoelastic strain error tolerance: Image: Creep/swelling/viscoelastic integration: Image: Explicit/Implicit Image: Explicit/Implicit

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

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

Edit Step	Edit Step
Name: Step-3 Type: Coupled temp-displacement Basic Incrementation Other Description: Remove contact pairs Response: Steady-state () Transient Time period: 0.1 NIgeom: On Edit	Name: Step-3 Type: Coupled temp-displacement Basic Incrementation Other Type: Automatic Fixed Maximum number of increments: 200 Initial Minimum Maximum Increment size: 0.1 1E-006 0.1 Max. allowable temperature change per increment: 100
Use stabilization with dissipated energy fraction < : 0.0002	Creep/swelling/viscoelastic strain error tolerance:
Include creep/swelling/viscoelastic behavior	Creep/swelling/viscoelastic integration: Explicit/Implicit Explicit

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

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

Edit Step	Edit Step
Name: Step-4 Type: Coupled temp-displacement Basic Incrementation Other Description: Let workpiece cool down Response: Steady-state Transient Time period: 10000	Name: Step-4 Type: Coupled temp-displacement Basic Incrementation Other Type: Automatic Fixed Maximum number of increments: 200 Initial Minimum Maximum Isource 100 0.1 10000
NIgeom: On Edit Use stabilization with dissipated energy fraction	Increment size: 100 0.1 10000 Image: Max. allowable temperature change per increment: 100 Image: Creep/swelling/viscoelastic strain error tolerance:
Include creep/swelling/viscoelastic behavior	Creep/swelling/viscoelastic integration: Explicit/Implicit Explicit

E. MODULE \rightarrow INTERACTION

(a) To create surfaces for interaction

- From the main menu bar, select
 View→Assembly Display Options
- Click the Instance tab, toggle off the visibility of Workpiece-1, see Fig.E1. Click OK.
- From the main menu bar, select
 Tools→Surface→Create.
- 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**.
- 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**.
- 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.





(b) To create the interaction property

- From the main menu bar, select
 Interaction→Property→Create
- 2. Name it IntProp-1. Under **Type**, select **Contact**.
- 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

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

- From the main menu bar, select Constraint →Create
- Name the constraint Die-RigidBody. Under Type, select Rigid body.
- The Edit Constraint dialogue box appears (Fig.E6).
 Under Region type, choose Body(elements) and then pick the die region.
- 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.

Edit Cont	act Property	Fig.E5	X
Name: IntP	rop-1	0	
Contact Pr	operty Options		
Tangential	Behavior		
Heat Gene	ration		
<u>M</u> echanic	al <u>T</u> hermal		Delete
Tangentia	Benavior		
Friction fo	rmulation: Penalt	у	-
Friction	Shear Stress El	lastic Slip	
Direction	ality: 🔘 Isotropic	Anisotropic (Standard	only)
🔲 Use sli	p-rate-dependent	data	
🔲 Use co	ontact-pressure-de	ependent data	
📃 Use te	mperature-depen	dent data	
Number	of field variables:	0	
Frictic Coef 0.1	on f		
	ОК	Cance	1

Edit Constraint		×
Name: Die-RigidBody Type: Rigid Body	Fig.E6	
Region type	Region	Edit
Body (elements)	(Picked)	Clear
Pin (nodes)	(None)	
Tie (nodes)	(None)	
Analytical Surface	(None)	
Reference Point Point: (Picked) Edit		
Adjust point to center of r	mass at start of analysis.	
Constrain selected regions (coupled thermal-stress	to be isothermal analysis only)	
ОК	Cancel	

(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.

	Name	Initial	Step-1	Step-2	Step-3	Step-4	
~	CONVECT					Created	
~	INTER-H	Created	Propagated	Propagated	Inactive	Inactive	
\checkmark	INTER-V	Created	Propagated	Propagated	Inactive	Inactive	
Step) procedure:	Coupled tem	p-displacement		F	ig.E7	
Step	procedure: raction type:	Coupled tem Surface film	p-displacement condition		F	ig.E7	
Step Inte Inte	procedure: raction type: raction status	Coupled tem Surface film :: Created in th	p-displacement condition is step		F	ig.E7	

- Thermal film interaction
- 1. From the main menu bar, select Interaction \rightarrow 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.
- In the Edit Interaction dialogue box, enter Film coefficient as 10 (W m⁻² K⁻¹) and Sink temperature as 20 (°C).

Region Selection	
Eligible Sets Surfaces below may contain faces.	Fig.E9
Name	Туре
Surf-die-Contact	Surface
Surf-workpiece-Convect	Surface
Surf-workpiece-Horizontal	Surface
Surf-workpiece-Vertical	Surface
Highlight selections in viewport	
Continue	Cancel

Create Interaction	<u> </u>			
Name: CONVECT				
Step: Step-4	Fig.E8			
Procedure: Coupled temp-displacement				
Types for Selected Step				
Surface-to-surface contact (Standard)				
Self-contact (Standard)				
Surface film condition				
Surface radiation to ambient				
Concentrated film condition				
Concentrated radiation to ambient				
Continue	Cancel			



86

- Mechanical interactions
- 1. From the main menu bar, select Interaction \rightarrow Create
- 2. Name it INTER-H. Select Step: Initial and Surface-tosurface 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.
- 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).
- 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



Li Edit Interaction					
Name: INTER-H					
Type: Surface-to-surface contact (Standard) Fig.E12					
Step: Initial					
Master surface: Surf-die-Contact					
Slave surface: Surf-workpiece-Horizontal					
Sliding formulation: Finite sliding Small sliding 					
Discretization method: Node to surface					
✓ Exclude shell/membrane element thickness					
Degree of smoothing for master surface: 0.48					
Use supplementary contact points: Selectively Never Always 					
Contact tracking: Two configurations (path) Single configuration (state) 					
Slave Node/Surface Adjustment Surface Smoothing Clearance					
No adjustment					
Adjust only to remove overclosure					
Specify tolerance for adjustment zone: 0					
Adjust slave nodes in set:					
Contact interaction property: IntProp-1 Create					
Options: Interference Fit					
Contact controls: (Default)					
OK Cancel					

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 \rightarrow 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.

Name	Initial	Step-1	Step-2	Step-3	Step-4
✓ Disp-BC-1		Created	Propagated	Propagated	Propagated
✓ Disp-BC-2		Created	Propagated	Propagated	Propagated
✓ Disp-BC-3		Created	Inactive	Inactive	Inactive
✓ Disp-BC-4			Created	Propagated	Propagated
✓ Temp-BC-1		Created	Propagated	Propagated	Propagated
Step procedure: Boundary conditio	Co on type: Dis	upled temp-disp placement/Rota	placement ation		

(i) Disp-BC-1

- 1. From the main menu bar, select **BC** \rightarrow **Create**
- 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.
- Set U1 = U2 = UR3 = 0 (To ensure that the die remains static throughout the simulation), see Fig.F3.

Edit Boun	dary Condition	×		
Name: Disp	Name: Disp-BC-1			
Type: Disp	Displacement/Rotation Fig.F.			
Step: Step	Step: Step-1 (Coupled temp-displacement)			
Region: (Picked)				
CSYS: (Glo	CSYS: (Global) Edit			
Distribution:	Uniform			
U 1:	0			
☑ U2:	0			
U R3:	0	radians		
Amplitude:	(Ramp)			
OK Cancel				



Module 4

(ii) Disp-BC-2

- From the main menu bar, select
 BC→Create
- Name it Disp-BC-2. Assign to Step: Step-1. Category:

$Mechanical \rightarrow Displacement/Rotation$

- 3. Pick the edge corresponding to the axis of the workpiece, see **Fig.F4**.
- Set U1 = 0 (To ensure that the workpiece remains axisymmetric throughout the simulation).

Fig.F4		Edit Boundary Condition
		Name: Disp-BC-2 Type: Displacement/Rotation Step: Step-1 (Coupled temp-displacement) Region: (Picked)
Pick his edge		CSYS: (Global) <u>Edit</u> Distribution: Uniform Ul: 0 U2:
R	P	UR3: radians Amplitude: (Ramp) OK Cancel
	W BP	

(iii) Disp-BC-3

- 1. From the main menu bar, select **BC** \rightarrow **Create**
- 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.



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

(iv) Disp-BC-4

- From the main menu bar, select
 BC→Create
- 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)

Pick this edge Fig.F6 Edit Boundary Condition Name: Disp-BC-4 Type: Displacement/Rotation Step-2 (Coupled temp-displacement) Step: Region: (Picked) CSYS: (Global) Edit... Method: Specify Constraints • Distribution: Uniform Ŧ 🔲 U1: VU2: -0.25 UR3: radians Amplitude: (Ramp) -ОК Cancel

(v) Temp-BC-1

- From the main menu bar, select
 BC→Create
- 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 the simulation since we are

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
- 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.

	Fig.F7
•	Edit Boundary Condition Name: Temp-BC-1 Type: Temperature Step: Step-1 (Coupled temp-displacement)
Z	Region: (Picked) Distribution: Uniform 💌 Magnitude: 20 Amplitude: (Instantaneous) 💌
	OK Cancel
Pick RP	

G. MODULE \rightarrow MESH

(a) To mesh the workpiece:-

- 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**.
- From the main menu bar, select Mesh→Controls. Use
 Quad elements and apply Structured (green) technique (Fig.G2).
- 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.
- From the main menu bar, select Mesh→Mesh
 Instance. Select the workpiece instance to generate the mesh, it should resemble Fig.G4.

Element Library	Family	
🖲 Standard 🔘 Explicit	Cohesive	
	Coupled Temperature-Displacement	ľ
Geometric Order	Gasket	
🖲 Linear 🔘 Quadratic	Heat Transfer	
Quad Tri		
Element Controls	Fig.G3	
Hybrid formulation	119.00	
Reduced integration	1	
Hourglass stiffness:	🔘 Use default 🔘 Specify	
Second-order accuracy:	Yes 🛞 No	
Distortion control:	🔘 Yes 🛞 No	
Leng	gth ratio: 0.1	
Hourglass control: 🔘	Enhanced 🔵 Relax stiffness 🔘 Stiffness 🔘 Viscous 🔘 Combined	
	Stiffness-viscous weight factor: 0.5	
Displacement hourglass	s scaling factor: 1	
CAX4T: A 4-node axisyn	nmetric thermally coupled quadrilateral, bilinear displacement and temperature.	
ator. To coloct an elemen	nt chang for maching	
select "Mech-> Con	nt snape for mesning, strols" from the main menu bar	



Mesh Controls		22
Element Shape	l-dominated 🔘 Tri	Fig.G2
 Technique As is 	Algorithm Options	sition Tip
Free Structured		
Sweep		
O Multiple	Redefine Region Corners	
ОК	Defaults	Cancel



91

(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.
- The generated mesh of the assembly is shown in Fig.G6.



H. MODULE \rightarrow 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 \rightarrow VISUALIZATION

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



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.