Final Presentation

Linh Vu – Technical Intern (Summer 2023)

[redacted] – Supervisor (TF Team)

08/31/2023

mailinhvu912@gmail.com or lvu@stanford.edu

Commonwealth Fusion Systems

Overview

- Problem
  - [redacted]
- Project Goal: Implement a solder model in Ansys
  - ✔ Create a plastic flow equation based on the experimental data of 60Sn40Pb or 63Sn37Pb solder.
  - ✔ Create an UPF usermat.F file containing the equation.
  - [redacted]

1. Gather experiment data:

σ_y and E of $Sn_{60}Pb_{40}$ at 4K and 77K

Table 2 Mechanical properties of lead/tin solders

% Sn	Temperature (K)	σ_u(MPa)	σ_y(MPa)	Elongation %
100	[redacted]			
60				
50				
40				
0				
100				
60				
50				
40				
0				

%Sn	Temp [K]	Young's Modulus (at 0.2% strain) [GPa]
[redacted]		

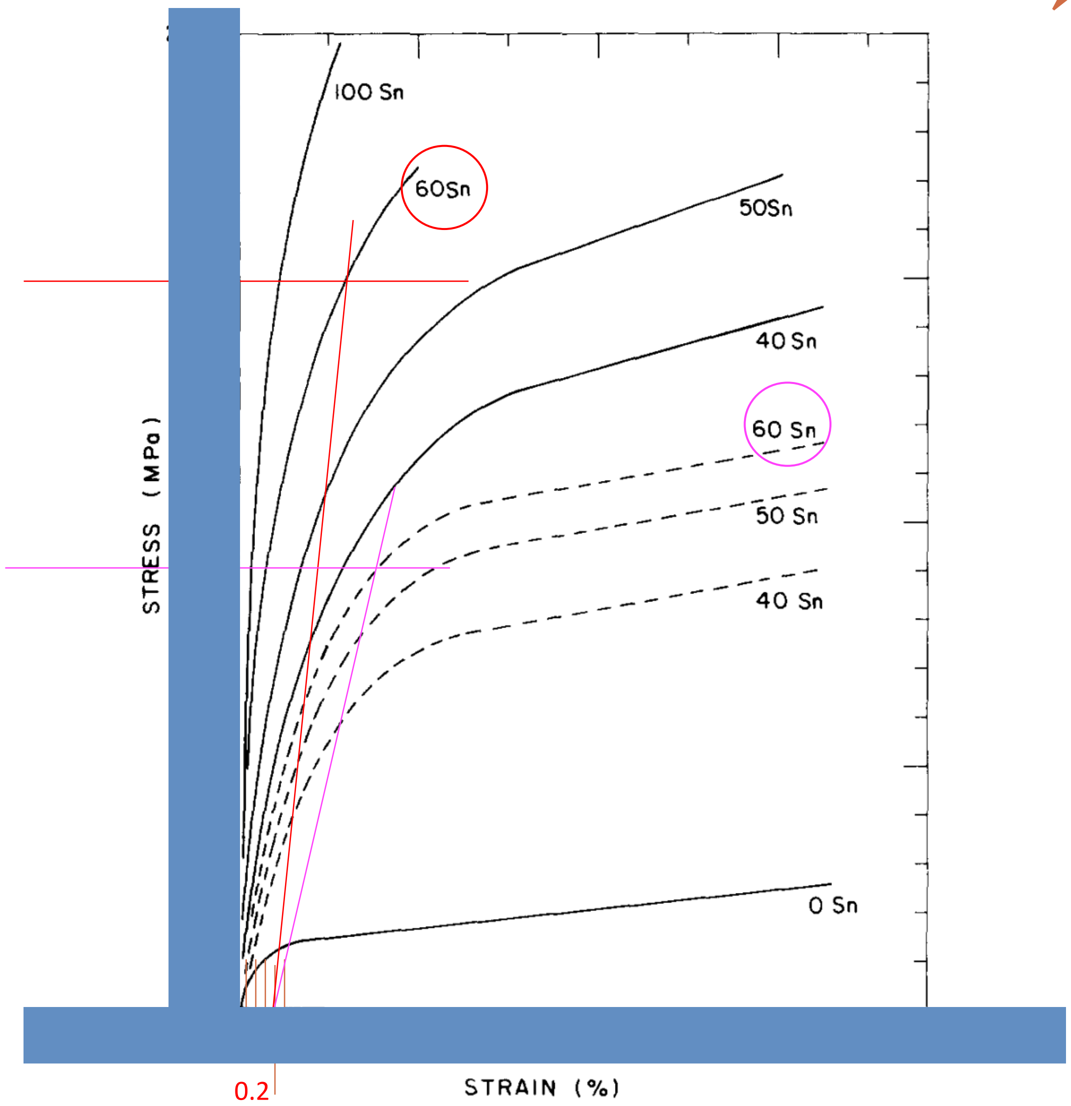


Figure 3 Stress–strain behaviour of various solders at 77 and 4.2 K. ——, 4.2 K; - - - - -, 77 K

1. Gather experiment data:

E of $Sn_{63}Pb_{37}$ at 233K → 398K

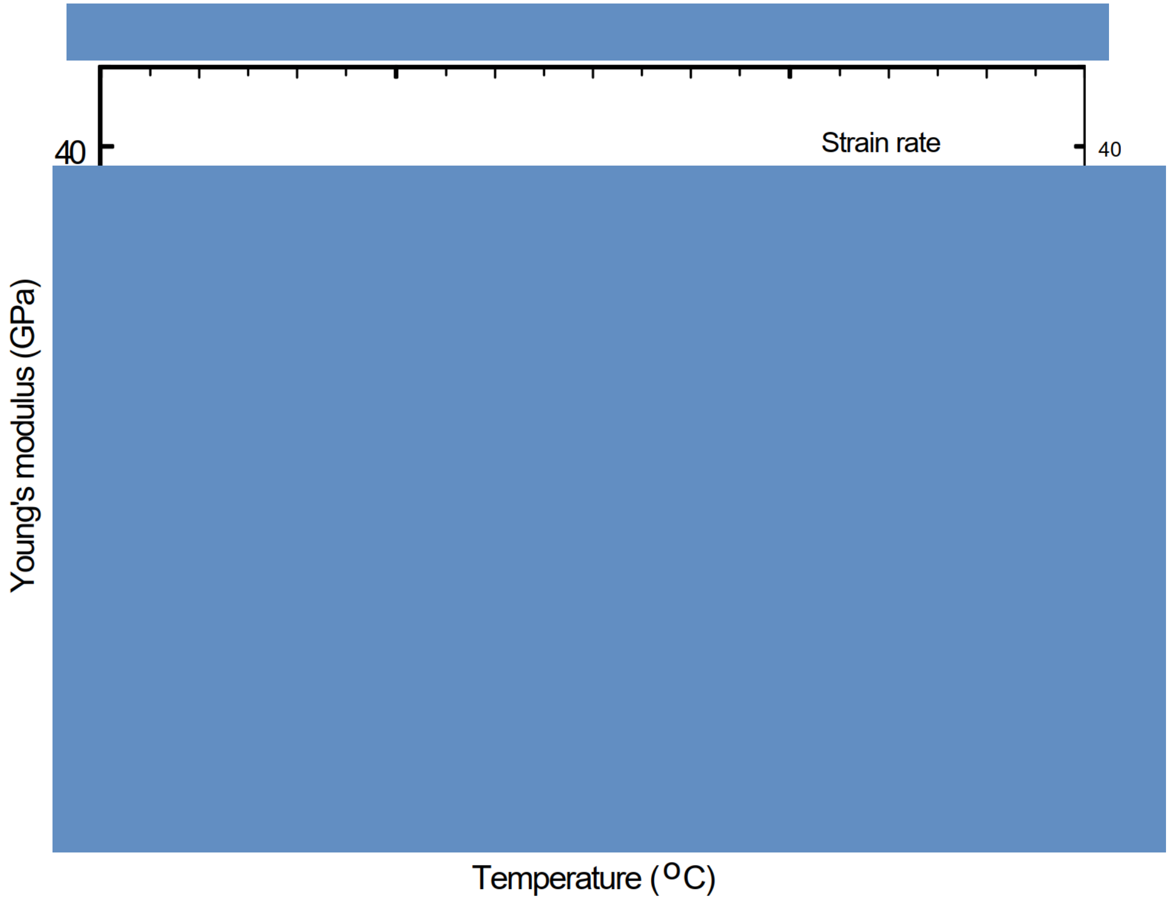


Fig. 3 Effect of temperature on Young's modulus

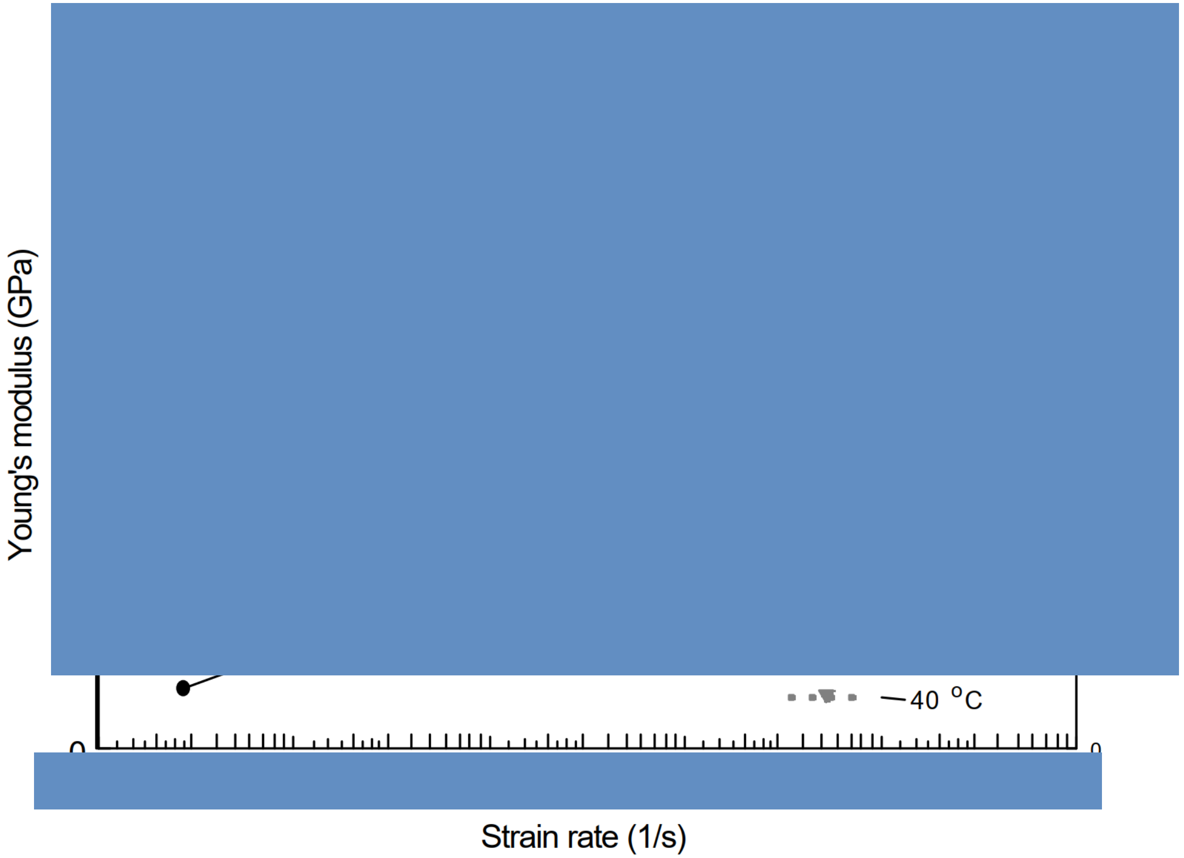


Fig. 4 Effect of strain rate on Young's modulus

1. Gather experiment data:

σ_y of $Sn_{63}Pb_{37}$ at 233K → 398K

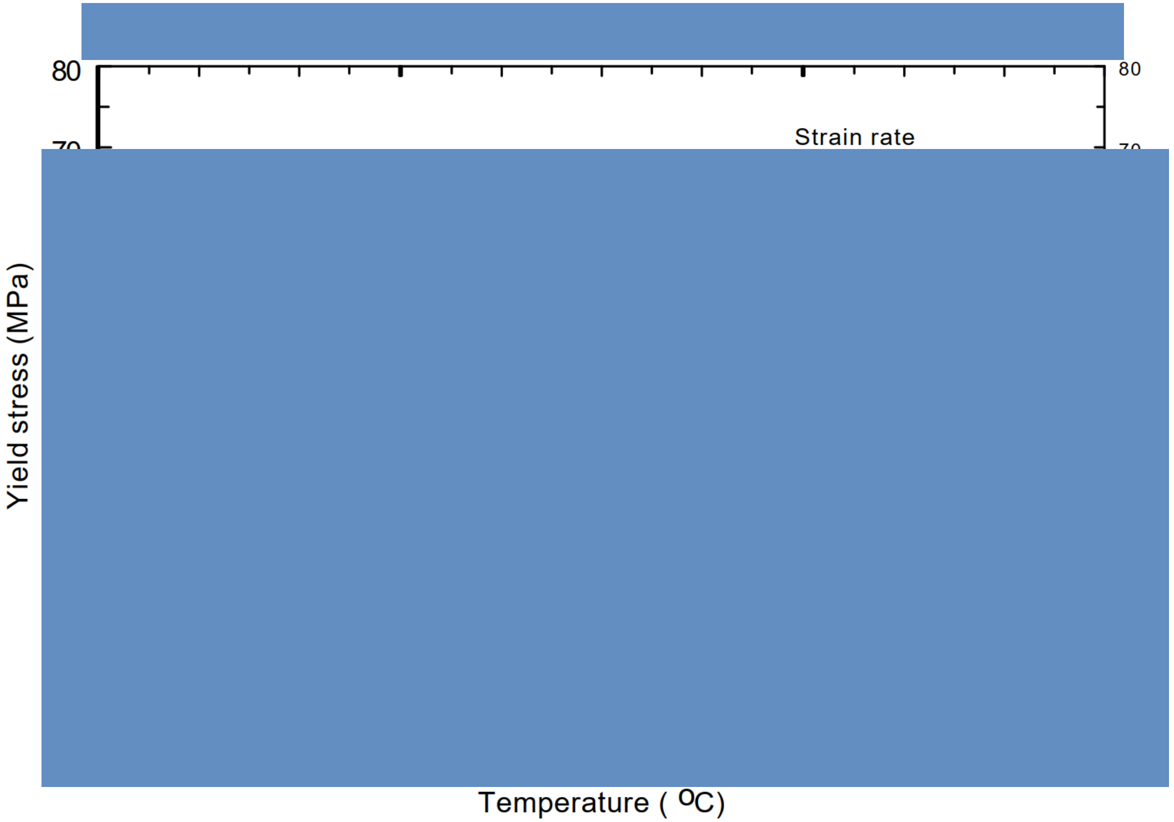


Fig. 5 Effect of temperature on yield stress

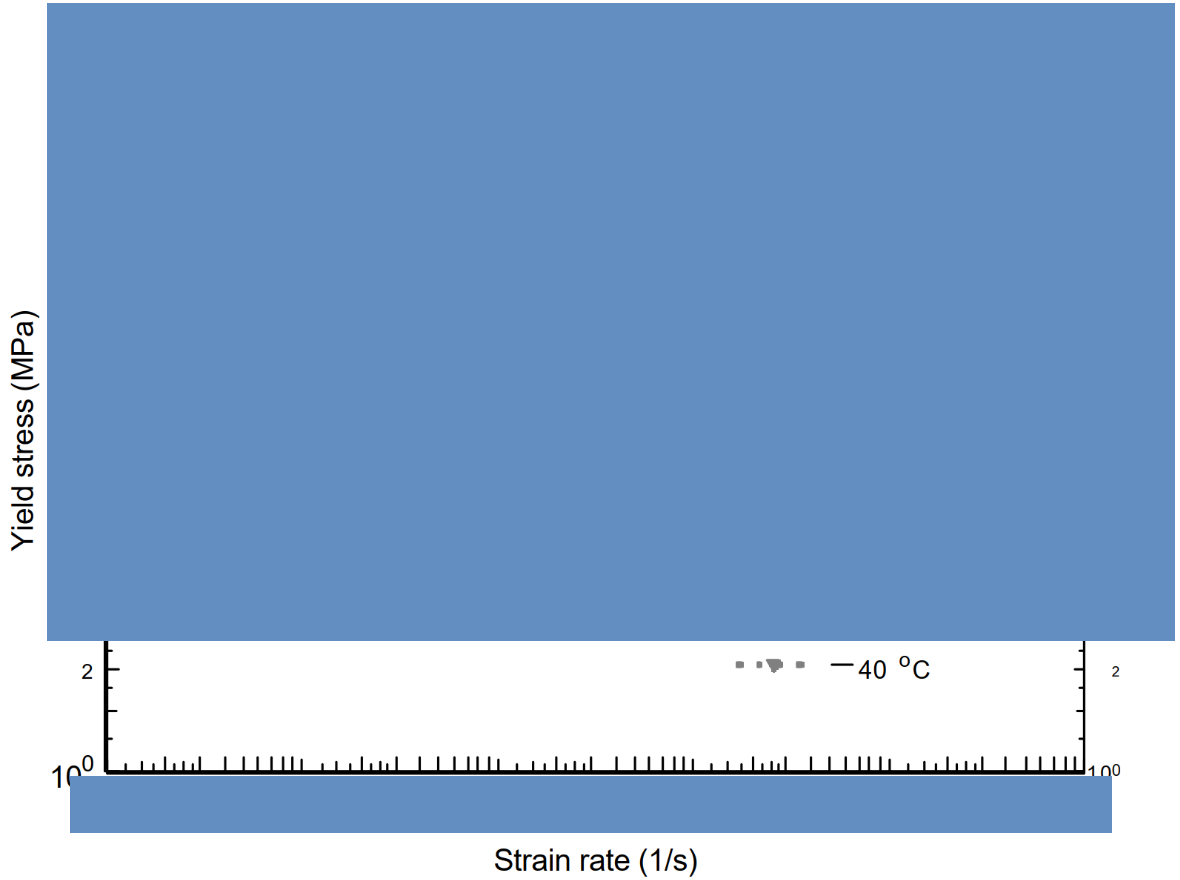


Fig. 6 Effect of strain rate on yield stress

1. Gather experiment data:

σ_y and E of $Sn_{60}Pb_{40}$ / $Sn_{63}Pb_{37}$ at 4K → 398K

Chosen for general material properties in Ansys

T [K]	$\dot{\varepsilon}$ = 2.78E-5 [1/s]		$\dot{\varepsilon}$ = 2.78E-4 [1/s]		$\dot{\varepsilon}$ = 2.78E-3 [1/s]		$\dot{\varepsilon}$ = 2.78E-2 [1/s]		$\dot{\varepsilon}$ = 2.78E-1 [1/s]		
	E [GPa]	σ_Y [MPa] at ε = 0.2%	E [GPa]	σ_Y [MPa] at ε = 0.2%	E [GPa]	σ_Y [MPa] at ε = 0.2%	E [GPa]	σ_Y [MPa] at ε = 0.2%	E [GPa]	σ_Y [MPa] at ε = 0.2%	Poisson's ratio

2. Re-create experiment data on plot via explicit integration method

- Loading conditions
  - Max true strain: $\varepsilon_{T_{max}}$ → Max engineering strain: $\varepsilon_{Eng_{max}}$
  - Strain rate: $\dot{\varepsilon}$
- Material properties
  - Young's Modulus: E
  - Initial deformation resistance: s_0
  - Initial plastic strain rate: $\dot{\varepsilon_0}$
  - Dimensionless constant: m, r
  - Saturation value of deformation resistance: s_{sat}
  - Hardening/softening constant: h_0

} Calibrate

- Output
  - Plastic strain rate: $\dot{\varepsilon_p}$
  - Deformation resistance: s

$$\varepsilon_{T_{max}} = \ln\left(1 + \varepsilon_{Eng_{max}}\right) \ (1)$$

$$\varepsilon_{n+1} = \varepsilon_n + \frac{\varepsilon_{Eng_{max}}}{N} (2)$$

$$\varepsilon_{T_{n+1}} = \ln(1 + \varepsilon_{n+1}) \ (3)$$

$$dt = \frac{\varepsilon_{T_{n+1}} - \varepsilon_{T_n}}{\dot{\varepsilon}} (4)$$

$$t_{n+1} = t_n + dt (5)$$

$$\dot{\varepsilon_{p_{n+1}}} = \dot{\varepsilon_0} \left(\frac{|\sigma_n|}{s_n}\right)^{1/m} \quad (6)$$

$$\varepsilon_{p_{n+1}} = \varepsilon_{p_n} + \dot{\varepsilon_{p_{n+1}}} \ dt \cdot sign(\sigma_n) (7)$$

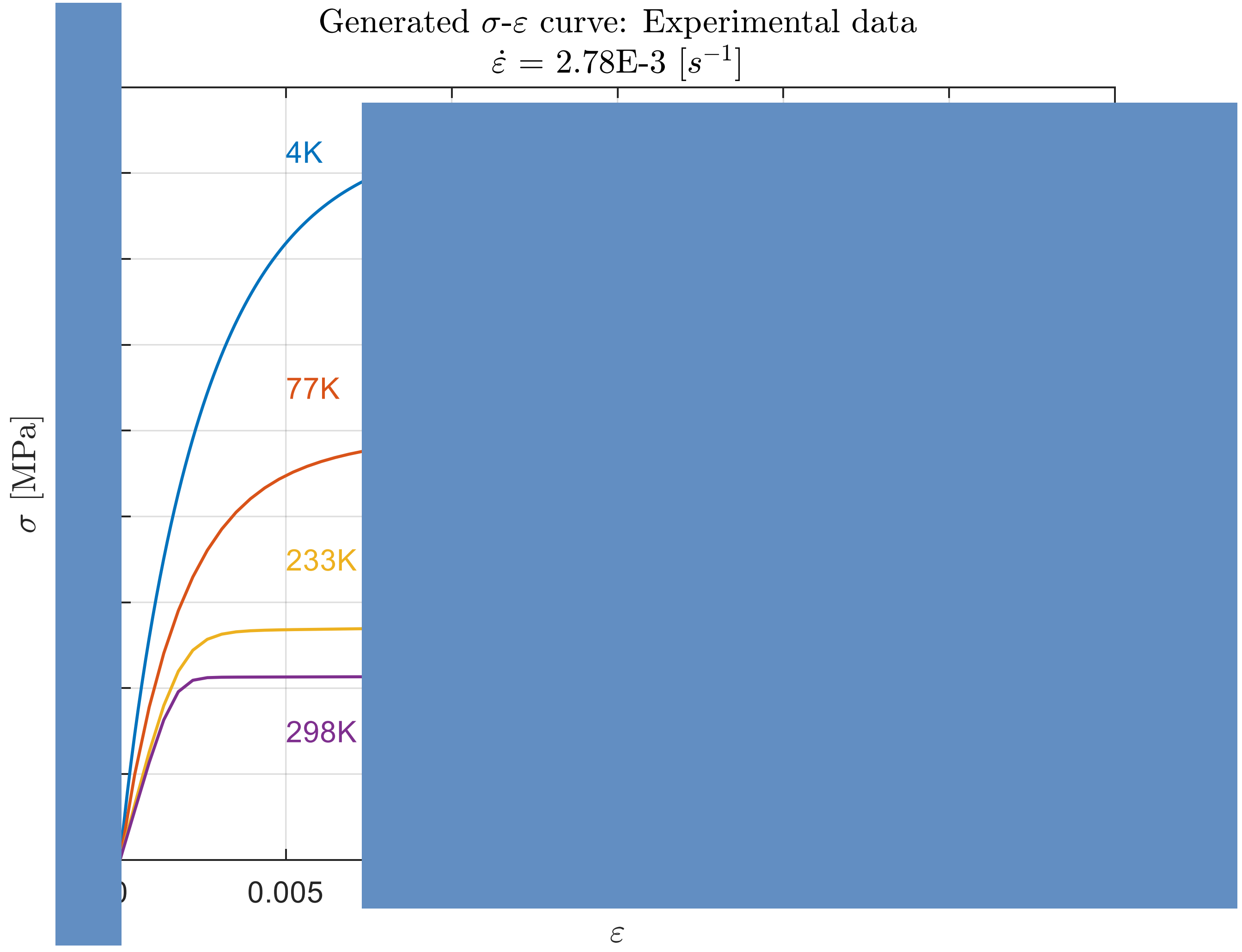
$$\dot{s_{n+1}} = h_0 \left(1 - \frac{s_n}{s_{sat}}\right)^r \dot{\varepsilon_{p_{n+1}}} (8)$$

$$s_{n+1} = s_n + \dot{s_{n+1}} \ dt (9)$$

$$\varepsilon_{e_{n+1}} = \varepsilon_{T_{n+1}} - \varepsilon_{p_{n+1}} (10)$$

$$\sigma_{n+1} = E \cdot \varepsilon_{e_{n+1}} (11)$$

2. Re-create experiment data on plot via explicit integration method



3. Create a temperature-dependent flow equation

- Initially, Anand's model is chosen: [Chen et al (2017) [3]]
  - No yield or loading conditions required.
  - The isotropic deformation resistance to plastic flow, s, is required
    → emphasizes T and $\dot{\varepsilon}_p$ sensitivity + strain hardening + …
  - Applicable to a wide range of temperature (i.e, 233K → 373K)
  - Is built-in in Ansys Engineering Data Toolbox.
  - However, the Arrehenius function $A \cdot e^{\frac{-Q}{kT}}$ is too steep for cryogenic temperature at 4K and 77K.

$$\dot{\varepsilon}_p = \underbrace{A \cdot e^{\frac{-Q}{kT}}} \cdot \sinh\left(\frac{\xi\sigma}{s}\right)^{1/m}$$

3. Create a temperature-dependent flow equation

- Busso's model is chosen: Chen et al (2017) [3]
  - Emphasizes σ dependent activation energy F_0 and Bauschinger effect.
  - Suggests the dependence of T and $\dot{\varepsilon_p}$ on shear stress.
  - However, the simplified version does not capture smooth elastic-plastic transition and isotropic hardening effect.

$$\sigma = \sigma_0 \, A \left\{ 1 - \left[\frac{\theta}{\frac{F_0}{R} \cdot \frac{1}{\ln\left(\dot{\varepsilon_0} / \dot{\varepsilon_p} \right)}} \right]^{1/q} \right\}^{1/p}$$

$$\dot{\varepsilon_p} = \frac{\dot{\varepsilon_0}}{exp\left\{ \frac{F_0}{\theta \cdot R} \left[1 - \left(\frac{\sigma}{\sigma_0 \cdot A} \right)^p \right]^q \right\}}$$

3. Create a temperature-dependent flow equation

- Fixed inputs
  - Temperature: θ
  - Gas constant: R
  - Yield stress at 0K: σ_0
  - Stress (from experiment data): σ
  - Plastic strain rate (from explicit integration, representing experiment data): $\dot{\varepsilon_p}$

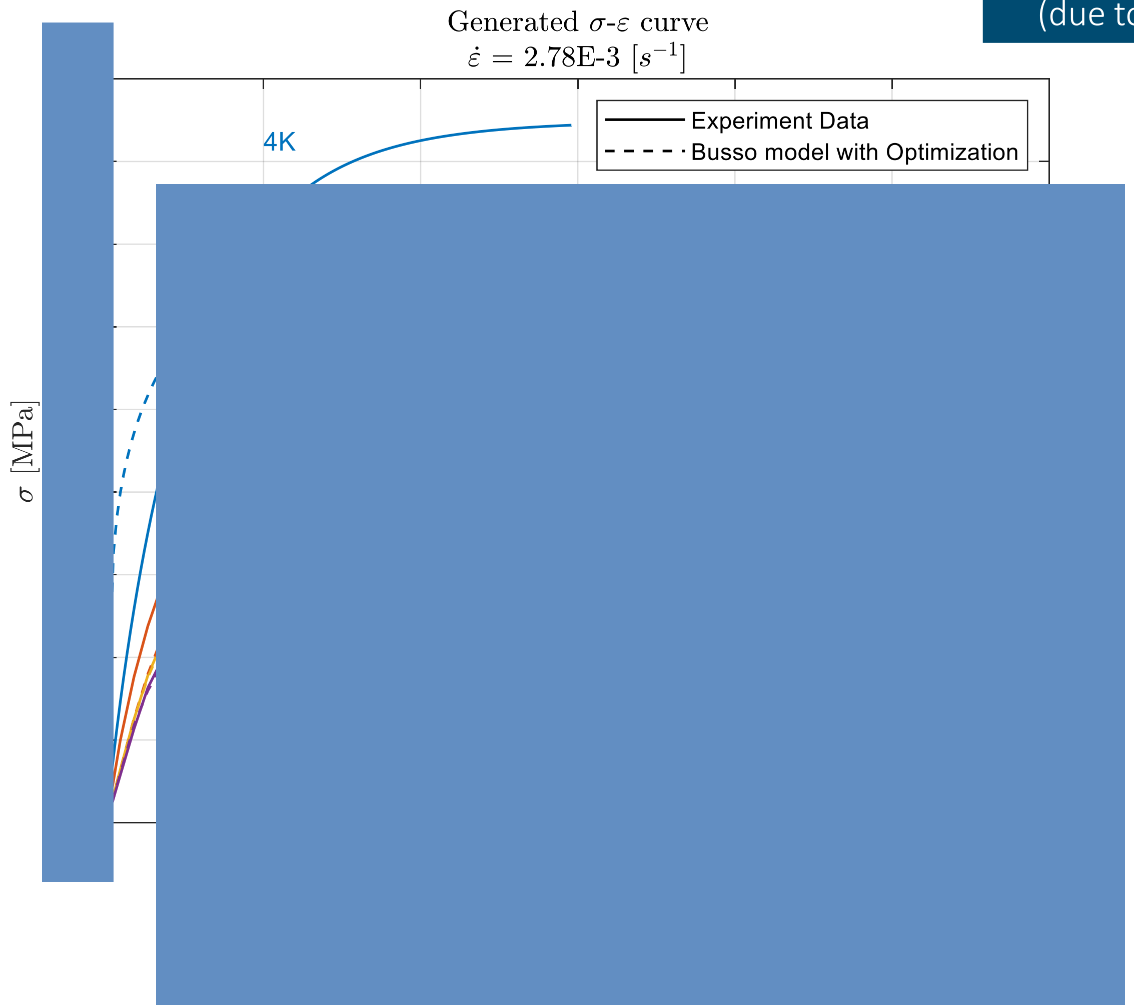
- Least-squared Optimization solutions
  - Free activation energy: F_0
  - Shear Modulus ratio: A
  - Initial strain rate: $\dot{\varepsilon_0}$
  - Material constants: p, q

Suggest initial values

$$\sigma = \sigma_0 \, A \left\{ 1 - \left[\frac{\theta}{\frac{F_0}{R} \cdot \frac{1}{\ln\left(\dot{\varepsilon_0} / \dot{\varepsilon_p} \right)}} \right]^{1/q} \right\}^{1/p}$$

```
diffexpr = sum((σcalculated - σexperimental).^2);

ssqprob = optimproblem('Objective',diffexpr);

opts =
optimoptions('fmincon','MaxFunctionEvaluations',
30000,'Algorithm','sqp');

[sol1,fval,exitflag,output] =
solve(ssqprob,x0,Options=opts);

resp1 = evaluate(diffun,sol1);
```

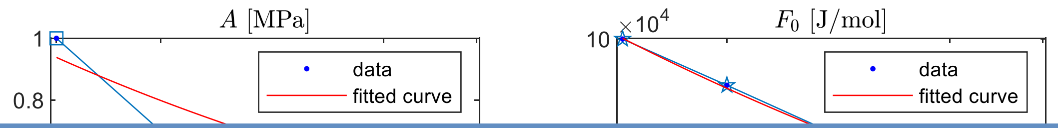
3. Create a temperature-dependent flow equation

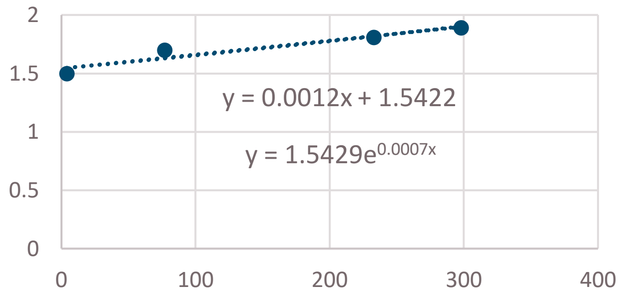


Sensitive (due to fmincon)

	OptimProblem initial suggestions	Results
4K	x0.A = x0.F0 = x0.edot x0.p = x0.q =	A F0 edot0 p q
77K	x0.A = x0.F0 = x0.edot x0.p = x0.q =	A F0 edot0 p q
233K	x0.A = x0.F0 = x0.edot x0.p = x0.q =	A F0 edot0 p q
298K	x0.A = x0.F0 = x0.edot x0.p = x0.q =	A F0 edot0 p q

3. Create a temperature-dependent flow equation



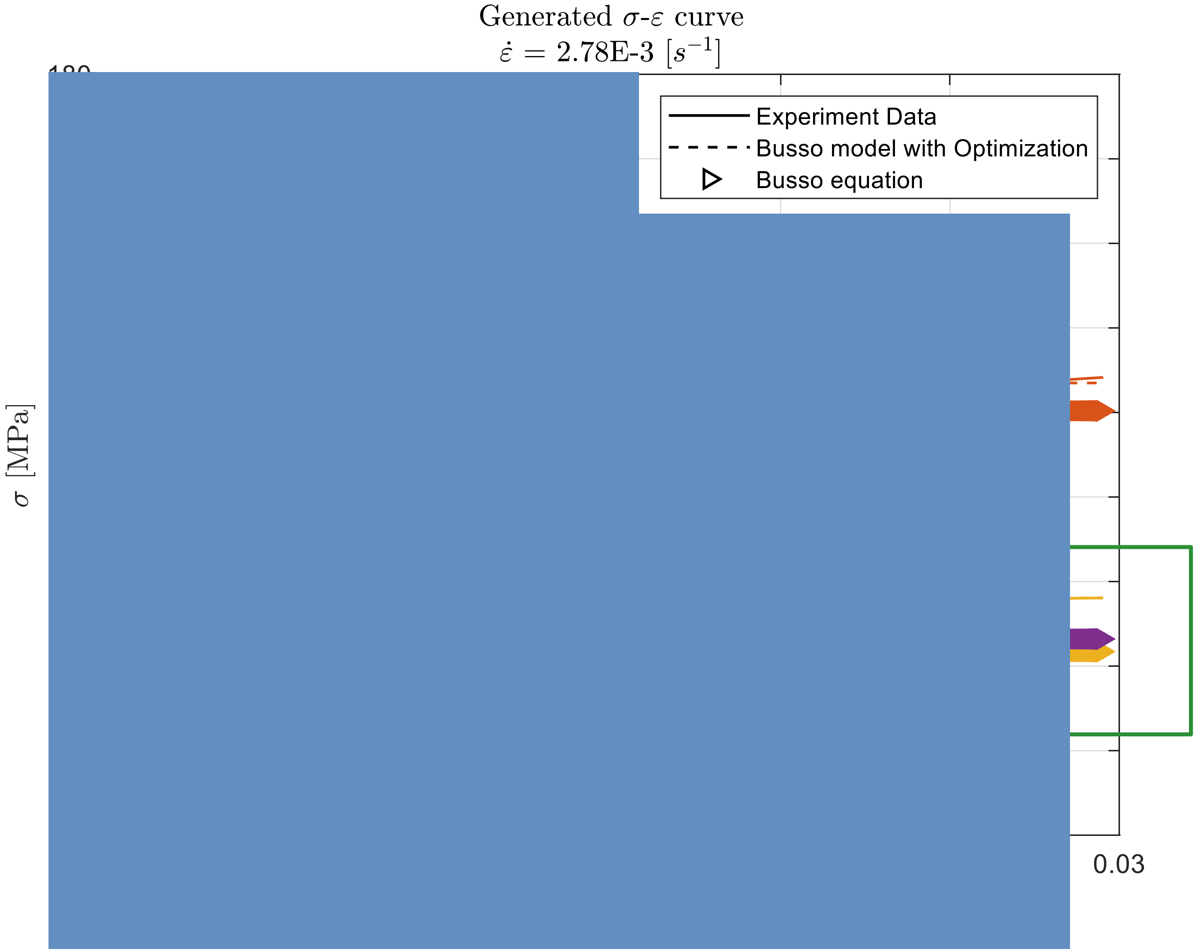


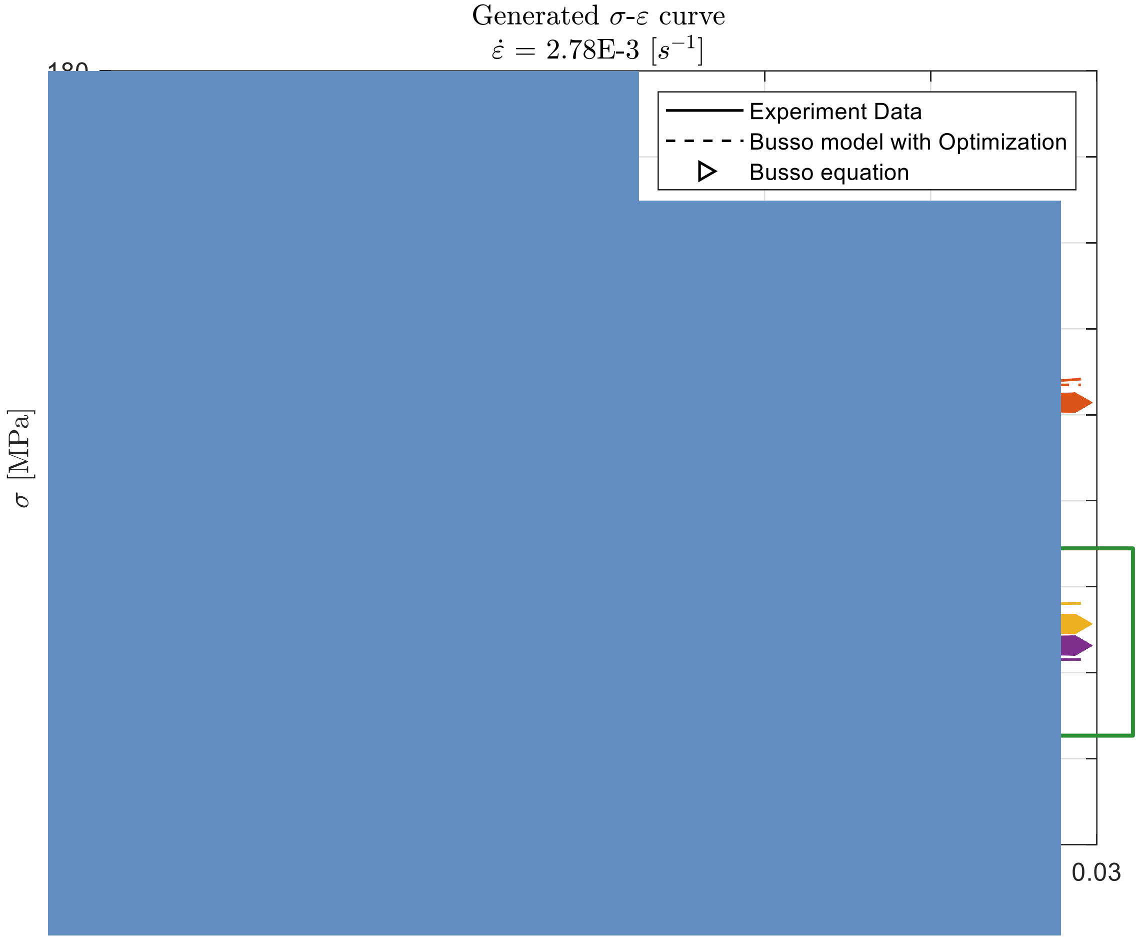
Fitted equations generated by Excel is ‘better’ by MATLAB

3. Create a temperature-dependent flow equation

Apply exactly the equations generated by Excel

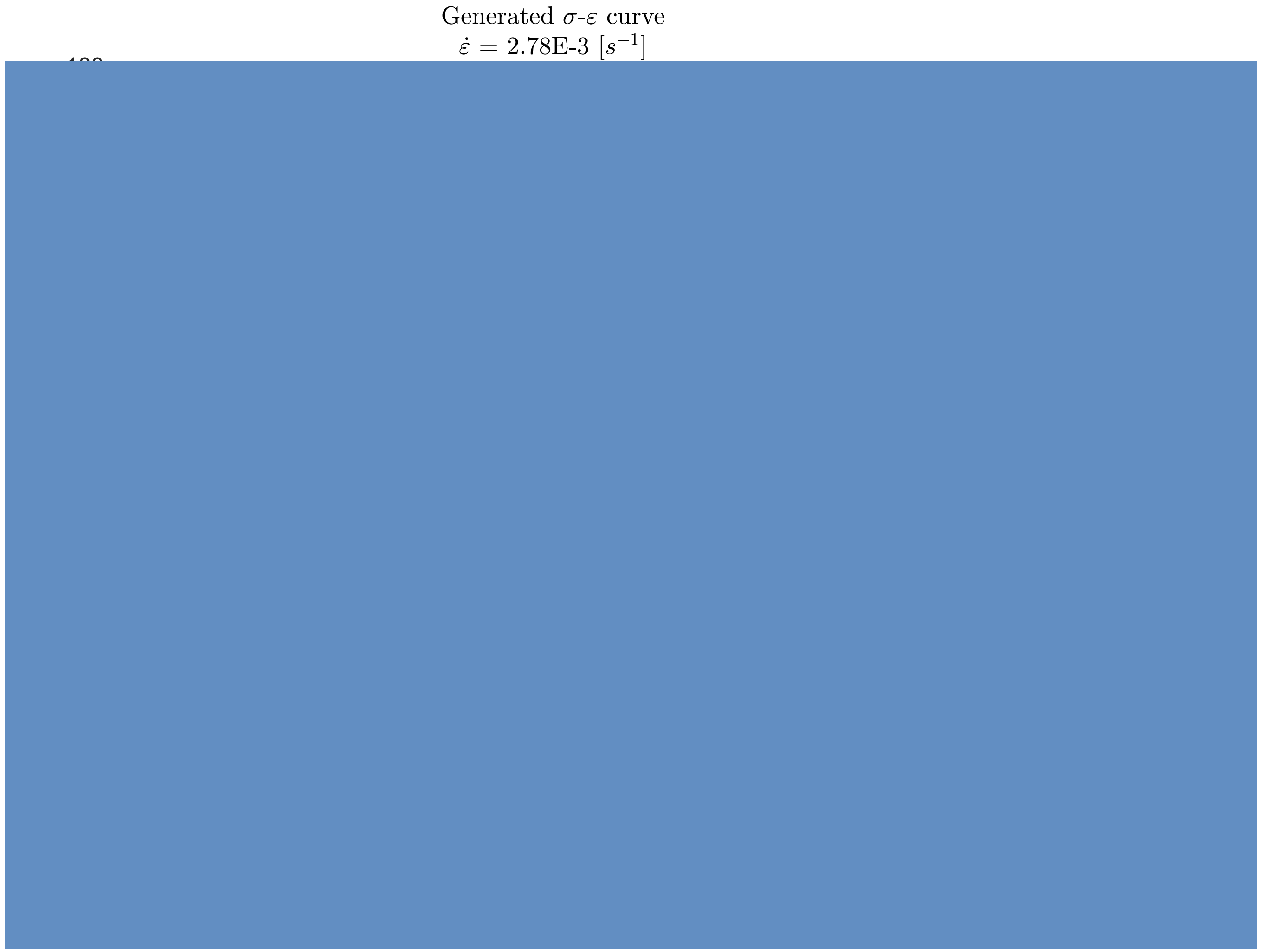
Adjust





3. Create a temperature-dependent flow equation

Final stress-strain curve



Final equation

$$\sigma = \sigma_0\ A\left\{1 - \left[\frac{\theta}{\frac{F_0}{R}\cdot\frac{1}{\ln\left(\dot{\varepsilon_0}/\dot{\varepsilon_p}\right)}}\right]^{1/q}\right\}^{1/p}$$

$$\dot{\varepsilon_p} = \frac{\dot{\varepsilon_0}}{exp\left\{\frac{F_0}{\theta\cdot R}\left[1-\left(\frac{\sigma}{\sigma_0\cdot A}\right)^p\right]^q\right\}}$$

4. Create an usermat.F file with the developed equation
4.5 Get UPF running in Ansys Mechanical

Getting started with APDL & UPF → Lots of errors! And lots of learning → Implement UPF & APDL in Ansys WB & Mechanical

Getting started with APDL & UPF

- Install Compilers
- Compile and Link UPF to Ansys
- Learn APDL & UPF Ansys courses
- Convert UMAT to USERMAT for a simple rate-dependent plasticity model with deformation resistance
- Create a single element with APDL

Lots of errors! And lots of learning

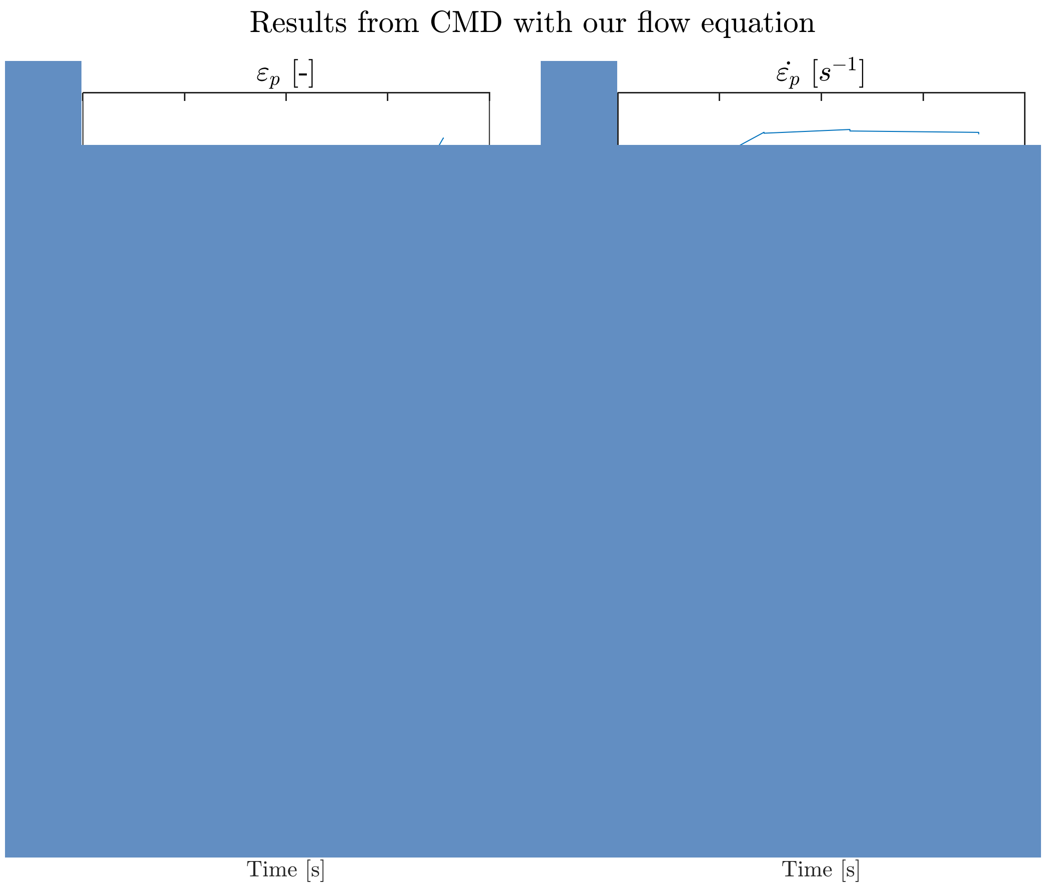
- Keyopt(2) that causes boundary condition errors
- Difference between the jacobian matrix position between ABAQUS and Ansys
- NROPT that proceeds for unsymmetric stiffness matrix
- Elastic → Plastic → Plastic with hardening → Plastic with our flow equation

Implement UPF & APDL in Ansys WB & Mechanical

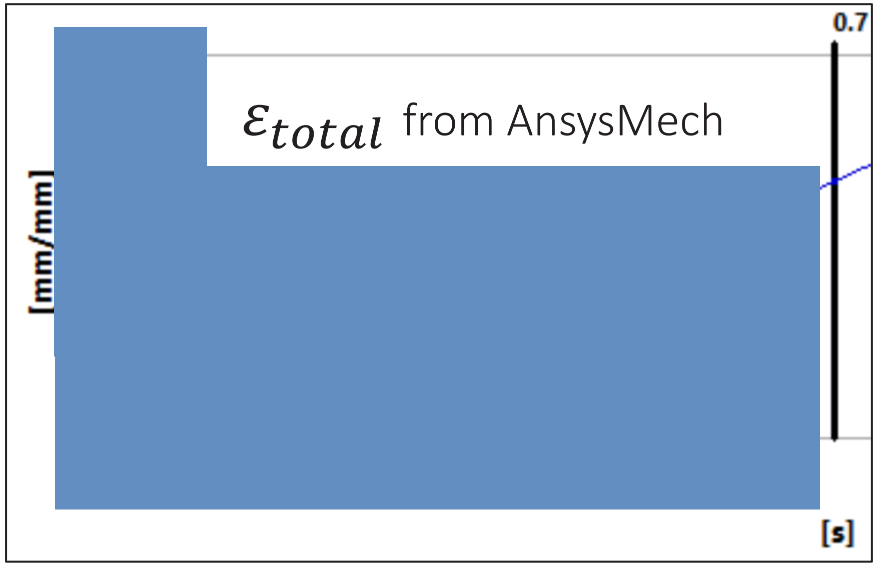
- Got identical results from commander and from Ansys Mechanical for deformation at 298K (flow equation is yet added)

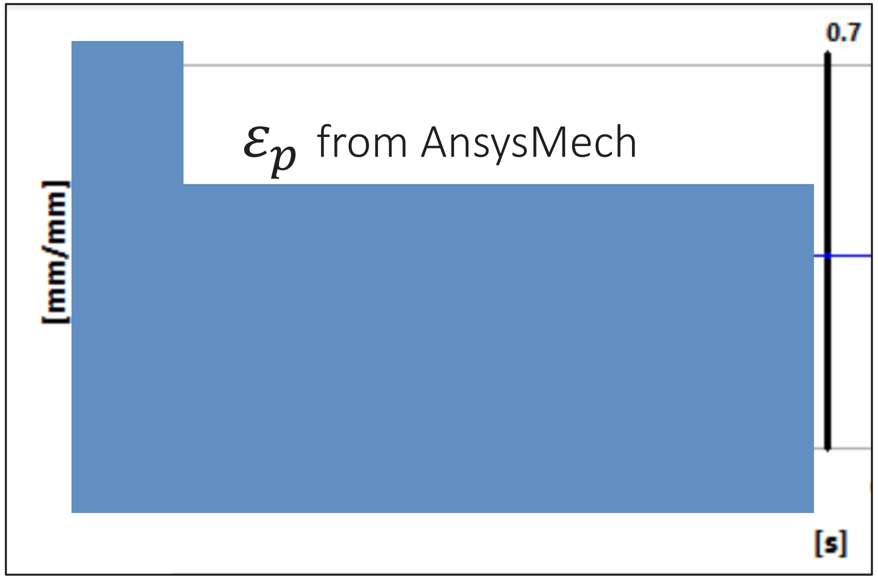
4. Create an usermat.F file with the developed equation
4.5 Get UPF running in Ansys Mechanical

However, results from the Usermat.F with flow equation added show different values from CMD and from Ansys Mechanical
→ Check Mechanical setting



$\neq$





4. Create an usermat.F file with the developed equation
4.5 Get UPF running in Ansys Mechanical

4. Create an usermat.F file with the developed equation
4.5 Get UPF running in Ansys Mechanical

Getting started with APDL & UPF → **Lots of errors! And lots of learning** → **Implement UPF & APDL in Ansys WB & Mechanical**

Getting started with APDL & UPF

- Install Compilers
- Compile and Link UPF to Ansys
- Learn APDL & UPF Ansys courses
- Convert UMAT to USERMAT for a simple rate-dependent plasticity model with deformation resistance
- Create a single element with APDL

Lots of errors! And lots of learning

- Keyopt(2) that causes boundary condition errors
- Difference between the jacobian matrix position between ABAQUS and Ansys
- NROPT that proceeds for unsymmetric stiffness matrix
- Elastic → Plastic → Plastic with hardening → Plastic with our flow equation

Implement UPF & APDL in Ansys WB & Mechanical

- Got identical results from commander and from Ansys Mechanical for deformation at 298K (flow equation is yet added)

↓

How to verify that this equation makes sense?

Conclusion

- What would I change?
  - Ask more
  - My knowledge and skills scope

- My personal achievements
  - Grateful for the precious learning opportunities (2nd time)
  - Genuinely more interested in FEA
  - Very meaningful to the design of my Master study and career

- Dedication
  - [redacted] – my manager
  - [redacted] – TF team
  - CFS