Cold bulk metal forming process with using mechanical type simulations.



1. Introduction

The aim of the task is to prepare a model of the process of the cylindrical sample compression test with dimensions: diameter=100mm and height=150mm. The sample is to be compress with two flat tools with diameter=200mm each. The lower tool is to be locked at all points of freedom, and the upper tool is to be moved from one base of the sample to the other until 20% deformation is achieved.

2. Material data and section creation.

The deformable material is an alloy of steel with specified mechanical properties, including the range of elastic-plastic deformation. Therefore, the material data required for the calculation in the ABAQUS application are material density, Young's modulus, Poisson's coefficient and stress-strain curve. To apply the material data correctly, it must be entered in "Materials" in the appropriate units. The model is created in mm, so the data must be entered into the application in accordance with the principle described in *Tab. 1*.

Tab. 1 Elastic properties of the steel used in the model.					
Units SI	Model in <i>m</i>	Model in <i>mm</i>			
Density	7950 $\frac{kg}{m^3}$	$7.95E - 9 \frac{tonne}{mm^3}$			
Young's module	210E9 Pa	210000 MPa			
Poisson's coefficient	0.3	0.3			

Tab. 1 Elastic properties of the steel used in the model.

Insert material data into ABAQUS in the "Materials" window (Fig. 1)

 A staduly circ corp [ricitpore i] 		
Eile Model Viewport View Material	+ Edit Material	×
	Name: Material-1	
	Description:	
Model Results Material Library	Material Benaviors	
😫 Model Database 🔤 🗘 🐑		
🖃 🎎 Models (1)		
<u>Model-1</u>		
Parts		
Materials	General Mechanical Thermal Electrical/Magnetic Other	*
R Section Materials		
+ Profiles		
🗉 🎎 Assembly		
teps (1) €		
Field Output Requests		
History Output Requests		
Ime Points ALE Adaptive Mask Constraints		
Contact Controls		
👔 Contact Initializations		
Contact Stabilizations		
Constraints		
Connector Sections		
j J Fields		
If plug-ins import this file Warning: Duplicate file name		
c:/Users/Work/abaqus_plugin		
If plug-ins import this fi.		
Warning: Duplicate file name c:/Users/Work/abagus plugin	OK	
c:/Users/Work/abaqus_plugin	Cancel	

Fig. 1 Material data insertion window.

The material data from the table are entered as **General->Density** and **Mechanical** ->Elasticity->Elastic (Fig. 2).

🜩 Edit Material	× 🖶 Edit Material ×
Name: Stal	Name: Stal
Description:	Description:
Material Behaviors	Material Behaviors
Density	Density
	Elastic
General Mechanical Ihermal Electrical/Magnetic Other	General Mechanical Ihermal Electrical/Magnetic Other
Density	Elastic
Distribution: Uniform 🗸 🥭	Type: Isotropic 🗸
Use temperature-dependent data	Use temperature-dependent data
Number of field variables: 0	Number of field variables: 0
Data	Moduli time scale (for viscoelasticity): Long-term 🗸
Mass	No compression
1 7.95E-9	□ No tension
	Data
	Young's Poisson's Modulus Ratio
	1 210000 0.3
OK Cancel	OK

Fig. 2 Density and elastic properties of the material.

The stress-strain curve is taken from the "*Steel_material_datas.xlsx*" file added into the course platform. In the *flow_stress* tab, we take, respectively, column D where the yield stress values are located and column C where the plastic strain values are located. Then insert the data from each column in "Edit Material" in the option **Mechanical->Plasticity->Plastic** (Fig. 3)

	Stal				
cripti	on:				
lateria	al Behaviors				
ensity	r				
astic					
lastic					
<u>i</u> enera	al <u>M</u> echanical	<u>Thermal</u> <u>Electri</u>	ical/Magnetic	<u>O</u> ther	
lastic					
arden	ing: Isotropic	\sim			▼ Suboptio
	strain rate depen	lant data			
] Use	strain-rate-depend	dent data			
] Use] Use	strain-rate-dependent	dent data endent data			
] Use] Use lumbe	strain-rate-depend temperature-dependence er of field variables	dent data endent data : 0			
] Use] Use lumbe Data	strain-rate-depend temperature-depe er of field variables	dent data indent data : 0			
] Use] Use umbe Data	strain-rate-dependent temperature-dependent er of field variables Vield Stress	dent data endent data : 0 • Plastic Strain			,
] Use] Use umbe Data	strain-rate-depend temperature-dependent er of field variables Yield Stress 553.4337483	dent data endent data : 0 V Plastic Strain 0			
Use Use Uumbe Data	strain-rate-depend temperature-dependent er of field variables Yield Stress 553.4337483 596.4848885	dent data endent data : 0 Plastic Strain 0 0.013154362			
Use Use Umbe Data	strain-rate-depend temperature-dependent er of field variables Yield Stress 553,4337483 596,4448885 626,818346	dent data endent data : 0 € Plastic Strain 0 0.013154362 0.026308725			
Use Use Uumbe Data	strain-rate-dependent temperature-dependent er of field variables Vield Stress 553.4337483 596.484885 626.818346 649.873675	dent data indent data Plastic Strain 0 0.013154362 0.026308725 0.039463087			
Use Use Data 1 2 3 4 5	strain-rate-depend temperature-depe er of field variables <u>Yield</u> Stress 553.4337483 596.4448885 626.818346 649.873675 668.3018505	dent data indent data i 0 ♥ Plastic Strain 0 0.013154362 0.026308725 0.039463087 0.05261745			
Use Use Data 1 2 3 4 5 6	Vield Stress 553,4337483 556,448885 626,818346 649,873675 668,3018505 6683,566236	dent data endent data 0 0 0 Plastic Strain 0 0.013154362 0.026308725 0.039463087 0.05261745 0.056771812			
Use Use Data 1 2 3 4 5 6 7	Vield Stress 553.4337483 596.44885 626.818346 649.873675 668.3018505 683.566236 683.566236	dent data indent data i 0 ♥ Plastic Strain 0 0.013154362 0.026308725 0.039463087 0.05261745 0.065771812 0.078926175			,
Use Use Data 1 2 3 4 5 6 7 8	strain-rate-depent temperature-depe er of field variables 593.4337483 596.4848885 626.818346 649.873675 668.3018505 683.566236 696.5490107 707.8180887	dent data endent data 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0			

Fig. 3 Plastic properties of the material.

The material data must then be assigned to the appropriate section defined in this case as homogeneous (Fig. 4).



Fig. 4 Creating sections and assigning material data.

3. Technical drawing

We make two technical drawings representing the sample and forming tools. The assembly to be done is shown, which consists of three parts (probe, lower tool and upper tool) in Fig. 5



Fig. 5 Model assembly.

The drawing of the forming tool is done once, as it will be double-loaded in the fold for the top and bottom tool respectively. The sample and tool dimensions are presented below.

 Sample dimensions (*Fig. 6*): R = 50mm and H = 150 mm (**3D->Deformable->Solid** ->Extrusion)



Fig. 6 Sample model.





Fig. 7 Creating a technical drawing of the sample in the ABAQUS environment.

• Tool dimensions (Fig. 7): R = 100 mm (**3D->Discrete Rigid->Shell->Revolution**)



Fig. 8 Upper and lower tool model.



Fig. 9 Create a tool drawing in the ABAQUS environment.

In order to be able to create any boundary conditions on tools that are of the "Rigid" type, a special reference point [RP] must be inserted in the "Tool" section. Use the option from the top bar to do this **"Tools"-> Reference Point** (Fig. 10). The location of such a point is arbitrary.





Fig. 10 Inserting a reference point.

Then assign the section created in point 3 to the sample in "Section Assignments" by following the commands given in the bottom bar (*Fig. 11*).



Fig. 11 Assigning sections to the sample

4. Define the time step.

The time step defines which type of simulation is to be performed. In this task we use the solver Implicit, so using the ABAQUS environment nomenclature we use the solver types Standard. Double click **Steps->Static**, **General (Step-1)** (Fig. 12)



Fig. 12 Create a time step in the ABAQUS environment.

🜩 Edit Step	🗙 🐥 Edit Step	×
Name: Step-1	Name: Step-1	
Type: Static, General	Type: Static, General	
Basic Incrementation Other	Basic Incrementation Other	
Description:	Type: Automatic Fixed	
Time period: 1	Maximum number of increments: 100000	
O Off (This setting controls the inclusion of nonlinear effects	Initial Minimum Maximum	
Nigeom: On of large displacements and affects subsequent steps.)	Increment size: 1E-05 1E-20 1	
Automatic stabilization: Specify dissipated energy fraction 🗸 : 0.0002		
Use adaptive stabilization with max. ratio of stabilization to strain energy: 0.05		
Include adiabatic heating effects		
OK Cancel	D) OK	Cancel

Fig. 13 Internal setting of the time step.

5. Preparation of the assembly.



Fig. 14 Assembly.

Now, with the help of previously prepared parts, you need to prepare the model assembly. The sample in the assembly is to be placed in the middle between the two tools. To do this, you will need to use the move and rotate options to create the required assembly part setting. So double click "Assemble->Instances". Load both "Probe" and "Tool" models using the "Shift" button. (Fig. 15).



Fig. 15 Loading parts for assembly

Now turn the "Tool-1" so that the possible contact surface is consistent with the lower surface of the sample.



Fig. 16 Select Rotate->Instance

Select the bottom tool



Fig. 17 Select the bottom tool

Select the axis of rotation against which you rotate the tool model by 90 degrees. • σ



Choose Rotate->Instance .



Fig. 18 Select the axis of rotation against which you rotate the tool model by 90 degrees.

Now we load the second tool "Tool-2", move it to the upper surface of the sample and then rotate it in the same way as it was done with the lower tool (Fig. 19).



Fig. 19 Loading the second tool with the Instances option.





Fig. 20 Loading the second tool with the Instances option.

6. Selection of boundary conditions.

In the example, the lower tool is to be locked in all degrees of freedom. Therefore, the "Encaster" option is set to the lower tool reference point in Step-1->BC-1->Mechanical ->Symmetry/Antisymmetry/Encaster.



Fig. 21 The first boundary condition is placed on the lower tool.

The upper tool moves by 30mm, deforming the sample by 20% from its height to Fig. 22.



Fig. 22 The second boundary condition is attached to the upper tool.

7. Define interactions.

In the "Initial" step, select **"General contact (Standard)".** This option allows to create an automatic contact in ABAQUS application. The method of establishing a "**General**" type of contact is presented on Fig. 23.

After selecting the option "General contact", select the way of contact between the tools and the sample. Here you must select the coefficient of friction and the value of the coefficient of friction through Contact->Mechanical->Tangential Behaviour->Friction Formulation (Penalty)->Friction Coeff (0.3).

8000 000	Part instances		1 550 3-	tastavas 🛛 🗗 • 🚹 🗇 🗇 🗇	Create Interaction P	ioperty 7	
			2 18.00	1.4	Name: IntProp-1	2	
	NET S Harner DEB	17 and 1 and 1	A state	manager II and Frances II and F	Tune	-	
Results	Madule Step: Initial in	these Passas	0 641 March	han X	type		
(Dalabere 🛛 🕄 🕄 🖒 🤅	Piecekee	Model Detabase	Non int		Contect	~	
Calibrations	Types for Selected Step	E Calinatar	Ige Greek	Loonland (Manufacelli	Film chedition		
Sections (1)	General contact (Mandard)	+ Polies	Step: Initial		Film Chaltion		
Profiles	will a furlace to surface contact (St	ndani) 0 👪 Assentiy	Centart Dam	-	Cavity radiation		
Assembly	f Bud costs	B Prite	Combanda included unfa	or pairs	Fluid cavity		
Position Constraints	Fluid exchange	* A feature	(I) SAF-44	h yell			
III Ja Features (7)	IFEM crack growth	dar Sets	O Selecte	disation pairs (Fluid exchange		
dar Ses	B Cyclic symmetry (Standard)	Converting of Converting	Excluded surfa	expany Nova 🎤	Acoustic impedance		
Ar Sufaces	Datic foundation	19.00	ring Fedures - "AB" include	call exterior faces and feature edges. It excludes	Incident wave		
Connector Assignments		Stages (2)	analoce she	sources, search represent, and removed powers.	Incluein wave		
S Steps (2)	10. 0	1	Attribute Ass	privati I	Actuator/sensor	100	
0+ basel		5.00	Carted Properties	Suface Centert	Leader and a second	•	
The Contract of Contract	X Castinue.	Cancel Bu Pres	efraid holds				
Be BCI	+ /	a St. Feb	Output Requests (1)		Continues	Cancel	
En Propried Fields	III 11	* <u>br</u> rea	ny Dulput Requests (1)	and and and a second	-		
S In Field Output Requests (1)	- **	A Edulation at an	~	C Edit Context Property	×		
= Per History Cutput Requests (1)	売末	- Edit interaction	^	Name Influent		um church arthrub	
ALE Adaptive Mesh Constraints	W X Is as the Control Manufacture of	Name: Int-1		Contact Property Dataons	5 \ "	mar Indhap-1	
3	1 March 1 Marc	Tana Canada a shaka (Shara dan	6	Distance I service and	<u> </u>	sense - related relations	
		Type: General contact (standard	•	Boot association and	1		
		Step: Initial			1		4
		Contact Domain				and the second second	
		Included surface pairs:		Mechanical Diserval Dectrical		Schanzal Dernal Excitical	
		included parter parte		Transfer Balancias		later. No options have been defined for this op	that property.
		(e) All" with self		targence territer		The Adapte defaults will be used.	
		O Selected surface pairs: N	Ne. J	Friction formulation: Penalty	H		
		Excluded surface pairs: None	,	Friction Shear Stress Electic Sky	Index balance		
		17AII' includes all exterior faces	and feature edges. It excludes	Use dip-rate-dependent data			
		analytical rigid surfaces, beam se	aments, and reference points.	Use contact-pressure-dependent data			
		Attribute Assignments		Die temperature dependent dats Nomber of Reld variables 0.02			
		Contact C. d.		friction			
		Properties Properties Form	ulation	Coeff 0.3			
		Global property assignment: In	tProp-1				
		Individual property assignment	: None				
		Initialization assignments: Nor	14				
		Stabilization assignments: Nor	1.7.38				
			(1997) (1997) (1997) (1997)				

Fig. 23 Creating General contact.

8. FEM discretization.

The discretization is done on both parts in parts in the module (Parts) and not in the module (Assembly). Please note that the student version does not allow for discrediting more than 1000 nodes per model.



Fig. 24 Sample FEM discretization.

💠 Abaqus/CAE 2019 - Model Database: C:\Users\Work\D	ysk Google\Dydaktyka 2020\2020 - Programy Symulacyjne MES - niestacjonarne\Classes E-Learning\speczanie_na_zimno.cae [Viewport: 1]	ø ×
Eile Model Viewport View Seed Mesh &	Ldaptivity Feature Iools Plug-ins Help א ?	_ 8 ×
] 🚰 🖬 🖶 🛔 🎒 🎒 💋	●●●●●●●●●●●●●●●●●●●●●●●●●●●●●●●●●●●●	
Model Results	Module 🛱 Mesh 🗸 Model 🖞 Model-1 😵 Object: 🔿 Assembly 🖲 Part 🛱 Tool	
Model Database	Image: Sector Sector Sing Controls Sing Controls Approximate global size [B] Image: Sector Sector Sing Controls Image: Sector Sector Sing Control Sector Image: Sector Sector Sing Control Sector Image: Sector Sector Sing Control Sector Image: Sector Sector <t< th=""><th>× ×</th></t<>	× ×
< >	E Set the data using the Global Seeds dialog	ÓS SIMULIF

Fig. 25 Tool FEM discretization.

9. Create a task and perform calculations.

The last step is to prepare the input file (Job-1.inp) for Solver Standard. Double click on "Job" to enter the calculation settings.

 Abaqus/CAE 2019 - Model Database: C:\Users\Wor File Model Viewport View Job Adaptin 	KDysk Google/Dydsityka 2020/2020 - Programy Symulacyjne MES - niestacjonarne/Cli ity _Co-execution _Optimization _Tools _Plug-ins _Holp _ *? : (الله التي المنابع الله الله الله الله الله الله الله الل	 Ætit Job X Name: Job-1 Model: Model-1
Model Results	Module: Job Model: Model-1 Step: Step-1	Analysis product: Abaqus/Standard Description: Submission General Memory Parallelization Precision
	Source Model V Model-1	Job Type © Full analysis O Recover (Explicit) O Restart
Raptivity Processes Clobs Lutions Coptimization Processes		Run Mode Background Queue Use Type Schmitz Time
		Summellardy ○ Immediately ○ Wait hrs. min. ○ At \$
	Continue Cancel	OK

Fig. 26 Order calculations from the model.

Now you have to right-click and run the "Submit" option, which sends calculations to the solver that is installed locally.



Fig. 27 Starting calculations with the Submit option.

10. Visualisation of results.

If you click the right mouse button on the "Job-1" and enable the **Monitor** option, you can see the numerical values or the message **"Complete"** is written, it means that ABAQUS counted the simulation correctly. Errors or warnings may appear then **"Error"** message will appear then you need to see in the logs what the problem is and correct it according to the above instruction. If everything calculated correctly you can go to the results analysis by right clicking on "Job-1" and **"Results".** Clicking **"Plot Contours on Defromed Shape"** allows you to view the distributions of different calculated values such as stresses, deformations and displacements (Fig. 28).



Fig. 28 Results of calculations.