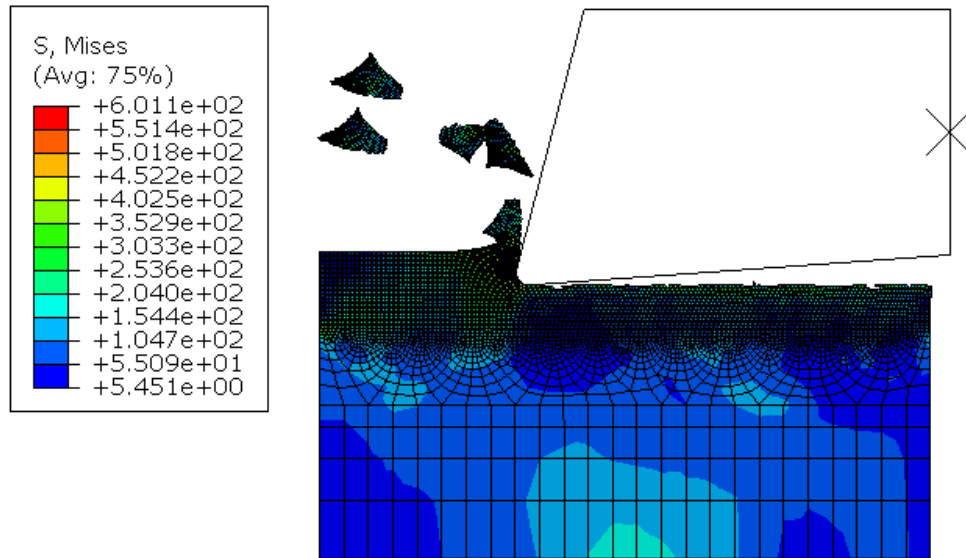


**ENGN 1750: Advanced Mechanics of Solids**  
**Abaqus plasticity tutorial**  
**Prof. Allan Bower**



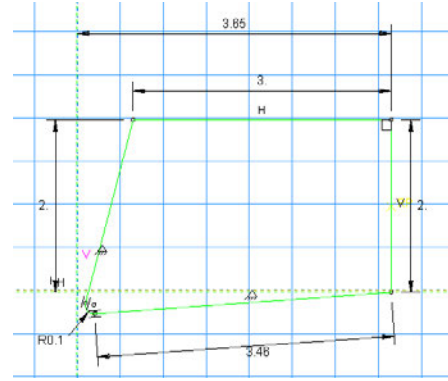
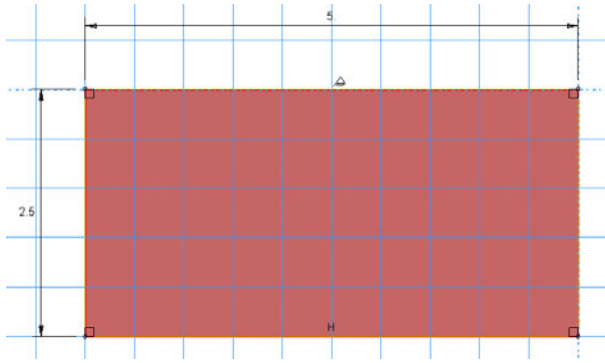
**Orthogonal machining model:** In this problem you will set up a simple 2D model of an orthogonal machining process (see the figure). The problem uses several advanced techniques in FEA:

- (1) Modeling contact
- (2) Modeling material failure, with element deletion
- (3) Using the 'Arbitrary Lagrangean-Eulerian' formulation available in ABAQUS to avoid problems caused by large distortion of the mesh.

The goal is to set up the model shown in the figure: a thin layer of material is removed from a workpiece (made from Al) by a rigid cutting tool.

For a unit system we will work with dimensions in mm; forces in N; stresses in MPa, and time in milliseconds (note that 1 m/s is 1 mm/millisecond).

6.1 Start by creating parts for both the workpiece and cutting tool. The workpiece is a 2D deformable part; the tool is a discrete rigid part with Wireframe base feature. Relevant dimensions are shown in the figure. To make it easier to position the parts in the assembly, place the top right corner of the rectangle representing the workpiece at the origin. You don't have to get the tool dimensions exactly the same as those shown in the figure. Note that the dimensions are in mm.



For mesh generation and adaptive meshing we will have to partition the workpiece as shown below. To do this, first partition the block down the middle – select the ‘partition by shortest distance between two points’



tool from the toolbar then select the midpoints on the two vertical sides of the workpiece. Next,

partition the top half of the block down the middle. To do this select the ‘Partition face by sketch’ tool, select the top half of the block in the viewport and press ‘Done’ then sketch a horizontal line down the center of the top half of the block. You should end up with a partitioned part that looks like the picture shown below.



Finally we need to add a reference point so we can apply boundary conditions to the tool. Make sure the tool is shown in the viewport, then select Tools > Reference point, and select the point midway down the back face of the tool (see the figure above).

6.2 In the Property module, create a new material called Al that will be assigned to the workpiece. [This publication](#) has material properties for a Johnson-Cook plasticity/damage model for a 5xxx series Al alloy. Enter values in MPa (this is convenient because  $1\text{N/mm}^2$  is the same as 1 MPa, so forces will be in Newtons)

- Young’s modulus  $E = 70\,000\text{ MPa}$
- Poissons ratio 0.3
- Density:  $2700\text{ kg/m}^3$  – but we have to be careful to enter its value in a consistent set of units. Convince yourself that if we choose to use N for force, mm for length, and milliseconds for time, we must enter density in  $\text{N milliseconds}^2/\text{mm}^4$ , which makes  $\rho = 2.7 \times 10^{-3}$  in our chosen unit system
- Plasticity – (select Mechanical>Plastic and use the Johnson-Cook hardening model in the drop-down menu. Then enter values of the parameters that define how the material hardens with strain and strain rate:  
 $A=277.3\text{MPa}$ ,  $B=307.93\text{MPa}$ ,  $n=0.69$ ,  $m=0$ , Melting  $T=0$ , Transition  $T=0$  (temp is neglected)  
 Select Suboptions, then select ‘Rate dependent’ and select the Johnson-Cook suboption. Enter  $C=0.0032$ , and Epsilon dot zero =  $10^{-6}$  (in milliseconds $^{-1}$ )

- Next, enter parameters that will control how the material begins to fail. In ABAQUS failure is modeled by making the material soften once the strain exceeds a critical value. To enter the parameters select Damage – select Mechanical > Damage for ductile metals > Johnson-Cook damage and enter for the constants:

$d1_1=0.211$ ,  $d2=3.9$ ,  $d3=-4.7$ ,  $d4=d5=0$

Finally, we need to enter a parameter that ABAQUS uses to control how elements are removed from the mesh after they fail. This is really a correction factor used during the simulation rather than a material property. It has two purposes: one is to make sure that the stress is reduced to zero gradually after an element fails; the second is to reduce sensitivity to mesh size that occurs in softening materials.

To do this select Suboption > Damage Evolution, and enter 0.01mm for the ‘displacement at failure’ (this makes ABAQUS remove an element when the plastic strain multiplied by the element size reaches 0.01mm)

(Read the [publication cited](#) or the ABAQUS theory manual for definitions of the constants)

Finally, create a homogeneous plane section with plane strain thickness of 5mm and assign it to the workpiece (the block).

### 6.3 Create an instance of both parts in the Assembly module.

6.4 In the Step module, create an Explicit Dynamic step with duration 17 milliseconds (enter 17 in the window, not 0.017, since we are working with millisecond time units). In the ‘Mass Scaling’ tab, check the radio button for ‘Use scaling definitions below’ and use Create... to define ‘mass scaling’ for the step. The purpose mass scaling is to increase the stable time-step for the analysis (we will discuss this in more detail later in the course). In the Edit Mass Scaling menu check the ‘Scale By Factor’ box and enter 10000 for the mass scaling (this is a rather too large –, but with smaller values you have to wait forever for the job to run).

Secondly, make sure the ‘Status’ variable is saved to the output database (otherwise elements don’t get deleted and the deformed mesh looks extremely scary). To do this select Output > Field Output Requests > Manager, then edit F-output-1. Expand the State/Field/User/Time list of variables and check the Status box and press OK at the bottom of the editor.

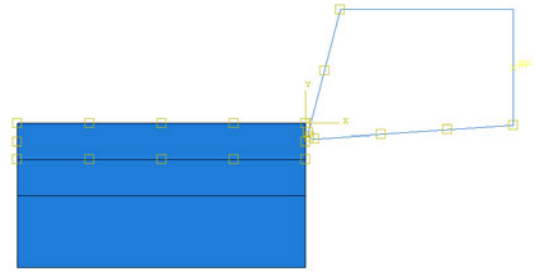
- ☐ DENSITYVAVG, Volume-averaged material density (Eulerian on
- ☒ STATUS, Status (some failure and plasticity models; VUMAT)
- ☐ EACTIVE, Status of the element

Next, use Tools > Set > Create to create a new set called Tool-Refpoint, and select the reference point on the tool. Then use Output> History Output Requests> Create... to create a new History Output request that will make ABAQUS save the forces and displacements of the reference point on the tool to the output database (select ‘Set’ for the domain, then you can select the Tool-Refpoint set; and check the boxes in the Output Variables list). Save the request....

Finally, the step menu also gives you an option to activate ALE remeshing. To do this select Other > ALE Adaptive Mesh Domain > Manager, and click on Step-1 and select ‘Edit’ in the domain manager. An Edit menu should open: select the ‘Use the ALE adaptive mesh domain below’ radiobutton, then press the arrow next to ‘Region’ and select the top rectangle of the part in the viewport. Accept all the defaults and press OK.

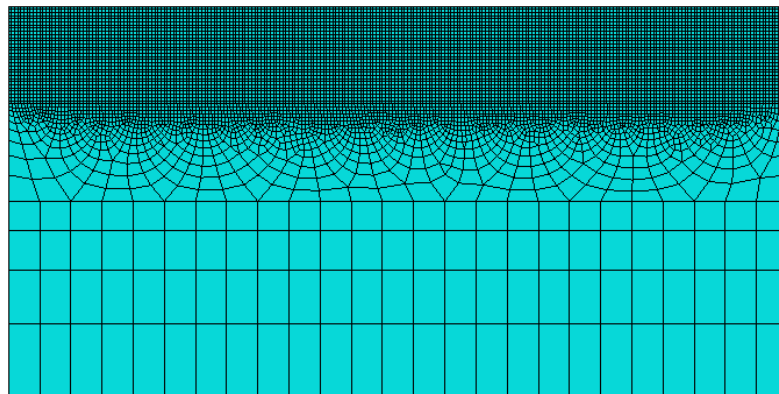
(Optional – if you want to make a smooth animation of the cutting process you will need to select more frequent outputs to the odb than the default – save every 0.5 secs or more frequently)

6.5 In the Interaction module: (a) create an interaction property with Hard normal contact and Penalty tangential contact with friction coefficient 0.1 (b) Create an Interaction (Interaction > Create, select Surface to Surface Contact (Explicit), then below the viewport select 'by angle' in the dropdown menu, and click on the fillet radius of the tool. The front face of the tool should be highlighted in the viewport. Select the color corresponding to the arrow on the outer side of the tool in the menu below the viewport. Then for the second surface select 'Node region' and select the rectangle at the top of the workpiece. The interaction should be highlighted as shown. Accept the default options in the interaction definition menu.



6.6 For boundary conditions: (1) Prescribe zero vertical displacement on the bottom of the block; (2) Prescribe zero horizontal displacement on the left edge of the bottom two partitions of the block; (3) Create a 'velocity/angular velocity' boundary condition for the reference point with a horizontal velocity of -0.25 mm/ms and zero vertical and angular velocity.

6.7 For meshing, use Mesh> Element Type... to assign Explicit Plane Strain elements – **check that the 'Element Deletion' option has the 'yes' radio button checked** - to the workpiece, then create a mesh like the one below (you can create it by seeding the various horizontal edges with different mesh densities. In the figure shown the top part has a uniform mesh size of 0.02mm). Seed the tool with a mesh size 0.02mm and mesh it.



6.8 Create a job and run it – it will take some time to complete – you can use Job> Monitor to track its progress.

As a solution to this problem create a plot of the horizontal force acting on the tool as a function of time.

**[10 POINTS]**