

ABAQUS VERSION 6.6-3

CUP FORMING – EXPLICIT ANALYSIS

PART → Criar a geometria e colocar os pontos de referência RP.

CHAPA

Part → Create → name: sheet → 3D → deformable → shell → revolution → → aprox. size 200 → continue → click create lines → segue prompts (parte de baixo da tela) → (0, 0) → enter → (50, 0) → enter → Esc → done → angle 90 → Ok.

PRENSA CHAPAS

Part → Create → name: Blank Holder → 3D → analytical rigid → revolved shell → aprox. size 200 → continue → click create lines → (39.83, -1.2) → enter → (39.83, 0) → enter → (70.18, 0) → enter → Esc → click create fillet → radius (2.0) → enter → click nas duas linhas → enter → Esc → Done → tools → reference point → select point.

PUNÇÃO

Part → Create → name: Punch → 3D → analytical rigid → revolved shell → aprox. size 200 → continue → click create lines → (0, 0) → enter → (25, 0) → enter → (25, -49) → enter → Esc → click create fillet → radius (5.0) → enter → click nas duas linhas → enter → Esc → Done → tools → reference point → select point.

MATRIZ

Part → Create → name: Die → 3D → analytical rigid → revolved shell → aprox. size 200 → continue → click create lines → (29.83, 15.5) → enter → (29.83, 0) → enter → (62.80, 0) → enter → Esc → click create fillet → radius (5.0) → enter → click nas duas linhas → enter → Esc → Done → tools → reference point → select point.

Part → fazer para todas as parte rígidas → engineering features → inertias (duplo click) →

** Para todas as partes rígidas clicar no RP.

name → point mass/ inertia → continue → select RP → done → Mass (magnitude) 7.6e-9 → Ok.

PROPERTY → Definindo as propriedades do material

Property → material → create → name: steel

General → density → 7.8e-9 (steel) [tone /mm³]

Mechanical → elasticity → elastic (ym = 210000 [MPa] → Pr = 0.3)

Plastic → Plastic

ys	ps
400	0.0
420	0.02
500	0.2
600	0.5

** Pegar valores à partir do pto. de escoamento

Section → create → name: blanksection → shell → homogeneous → continue → Shell tickness = 1.2 → thickness integration rule → simpson → integration points → 5 → ok

Assign → section → seleciona-se a chapa → done → blanksection (linhar a seção e a chapa) → ok.

ASSEMBLY → Montando as partes

Instance → create → sheet → ok

Instance → create → punch → ok

Instance → create → blank holder → ok

Instance → create → die → ok

Constraints (posicionando corretamente os componentes) → face to face → seguir prompts → click nas superfícies do punção e da chapa → se as setas estão na mesma direção click Ok, caso contrario flip e então Ok. (se as setas estiverem para baixo, os valores devem ter seus sinais invertidos, ou seja, para baixo será positivo)

Punção = 0.6 em y (selecione o punção e depois a chapa)

Matriz = -0.6 em y (selecione a matriz e depois a chapa)

Prensa chapas = 0.6 em y (selecione o B.H. e depois a chapa) – neste caso está em baixo da chapa.

**** Cuidado com os sinais**

Set → set das geometrias (para especificar loads e boundary conditions BC's)

Duplo click na árvore para criar

→ name: Chapa → continue → selecione a chapa → done

→ name: Matriz → continue → selecione RP → done

→ name: Prensa chapas → continue → selecione RP → done

→ name: Punção → selecione o RP → done

**** Deve estar selecionado o referential point (RP) e chapa.**

Surfaces → define as superfícies que entrarão em contato → Duplo click

name: BLH → continue → selecione a part → done → selecione a cor da parte de cima do blank holder

Die → continue → selecione a part → done → selecione a cor da parte de de baixo da matriz

Punch → continue → selecione a part → done → selecione a cor da parte de cima do punção

Lower blank to punch → continue → selecione a part → done → selecione a cor da parte de baixo do blank

Upper blank to die → continue → selecione a part → done → selecione a cor da parte de cima do blank

STEPS → Definição dos passos da simulação

Steps → create →

Nome: BLH comes up → general → dynamic explicit → continue
(description: push the blank holder towards sheet)

Time period: 0.01
 Nlgeon → On
 Mass scaling → use scaling definitions below → create → scale by factor → 60 → ok → ok

BLH force → apply blank holder force general → dynamic explicit → continue
 Time period: 0.01
 Nlgeon → On
 Mass scaling → use scaling definitions below → create → scale by factor → 30 → ok → ok

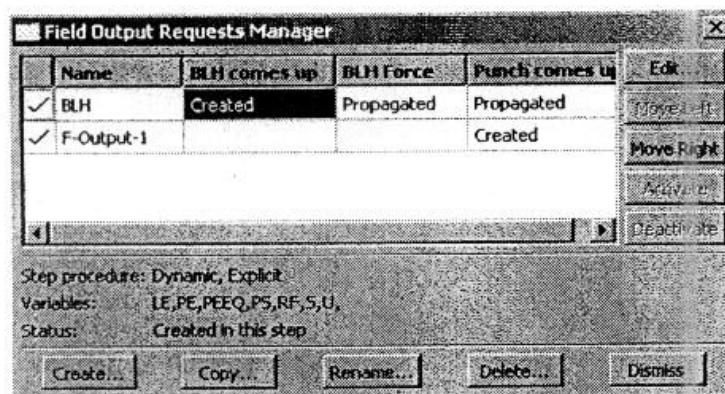
Punch comes up → Move the punch for top general → dynamic explicit → continue
 Time period: 0.008
 Nlgeon → On
 Mass scaling → use scaling definitions below → create → scale by factor → 30 → ok → ok

Fields output request (um geral é criado automaticamente) - fazer pelo menos 2.

F-Output 1 → punch comes up → continue → model → interval 20 → seleciona as 4 primeiras opções (não contact) → LE, PE, PEEQ, PS, RF, S, U → ok
 ou

F-Output 2 → punch comes up → continue → whole model → interval 20 → seleciona as 5 primeiras opções: A, CStRESS, LE, PE, PEEQ (maximum plastic strain), PS, RF, S, STH (espessura), U, V, VE, VEEQ, VS → ok

where: Spatial acceleration at nodes, contact stresses, logarithmic strain components, plastic strain components, equivalent plastic strain, reaction force, stress components, spatial displacement at nodes, spatial velocity at nodes.



History outputs requests (11)

H-output1 - criado automático – punção sobe → model → interval 200 → energy (deixa tudo selecionado)

Fazer os outros 10:

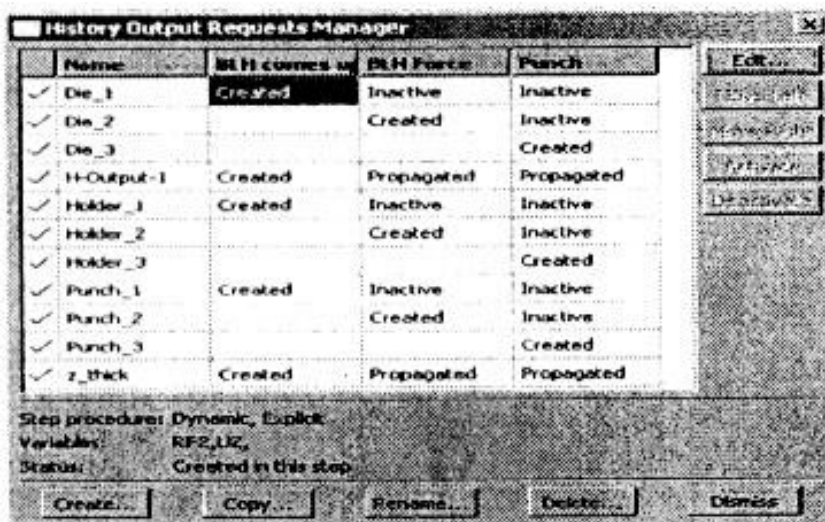
Chapa (lower blank) → punção sobe → continua → set → chapa → check box strain (importante é a PE, LE) → Interval 200

(quando peço stress (Misses), volume/thickness/coordinates STH (thickness) o abaqus não suporta o pedido de muitos dados – dá erro).

Die 1 → BLH comes up → continue → set → die → check box → displacement and force
 Die 2 → BLH force → continue → set → die → check box → displacements and force
 Die 3 → punch comes up → continue → set → die → check box → displacements and force
 For all interval 100, RF2, U2 (reaction force, spatial displacement).

Hoder 1 → BLH comes up → continue → set → holder → check box → displacement and force
 Hoder 2 → BLH force → continue → set → holder → check box → displacement and force
 Hoder 3 → punch comes up → continue → set → holder → check box → displacement and force
 For all interval 100 and RF2, U2

Punch 1 → BLH comes up → continue → set → punch → check box → displacements and force
 Punch 2 → BLH force → continue → set → punch → check box → displacements and force
 Punch 3 → punch comes up → s continue → set → punch → check box → displacements and force
 For all interval 100 and RF2, U2



INTERACTIONS

Interactions → property → create → name: friction punch and BH → contact → continue → mechanical → tangent behaviour → penalty → friction coefficient → 0.12 → mechanical → normal behaviour → default → Ok.

Interactions → property → create → name: friction for Die → contact → continue → mechanical → tangent behaviour → penalty → friction coefficient → 0.0225 → mechanical → normal behaviour → default → Ok.

Criar as interações:

Interactions → create → name: Holder - low blank → BLH comes up → surface to surface (explicit) → continue → clique na superfície rígida (parte de cima do holder) → done → escolha a cor → SURFACE segunda superfície → click na superfície → done → escolher a cor (bottom sheet) → penalty → contact interaction property → friction → Ok.

Interactions → create → name: Punch – low blank → punção sobe → surface to surface (explicit) → continue → clique na superfície rígida (parte de cima do punch) → done → escolha a cor → SURFACE segunda superfície → click na superfície → done → escolher a cor (bottom sheet) parte de baixo da chapa → penalty → contact interaction property → friction → Ok.

Interactions → create → name: Die - upper blank → BLH sobe → surface to surface (explicit) → continue → clique na superfície rígida (parte de baixo do holder) → done → escolha a cor → SURFACE segunda superfície → click na superfície → done → escolher a cor (upper sheet) parte de cima chapa → penalty → contact interaction property → friction for die → Ok.

LOAD

Primeiro fazer BC's (boundary condition) para cada uma dos Steps - constraints

BC's para o Step 1 (o step inicial não conta)

BC → create → Fix punch → holder comes up → mechanical → displacement/rotation → continue → select RP → done → restring todos os movimentos (tudo selecionado e zero) → amplitude → create → punch → smooth step → continue

Time Frequency (same time period of step)	Amplitude
0	0
0.008	1

→ amplitude → select punch (não instantaneous) → ok

BC → create → Fix blank holder → holder comes up → mechanical → displacement/rotation → continue → select RP → done → restringe todos os movimentos porém coloca $u_2 = 0.00012$, vai andar para cima- positivamente, (pequena pressão no BLH para causar um pequeno movimento para cima e fixar a chapa, serve para evitar vibração → amplitude → create → BLH → smooth step → continue

Time Frequency (same time period of step)	Amplitude
0	0
0.01	1

→ amplitude → select punch (não instantaneous) → ok

BC → create → Fix die → holder comes up → mechanical → displacement/rotation → continue → select RP → done → restring todos os movimentos (tudo selecionado e zero) BC → create → amplitude → instantaneous → ok

BC → create → Fix blank1 x-sym → holder comes up → mechanical → symmetric/antisymmetric/encastre → done → restring movimentos XSYMM ($u_1 = u_2 = u_3 = 0$) BC → create → ok (eixo z)

BC → create → Fix blank2 z-sym → holder comes up → mechanical → symmetric/antisymmetric/encastre → done → restring movimentos ZSYMM ($u_3 = u_1 = u_2 = 0$) → create → ok (eixo x)

BC's para o Step 2:

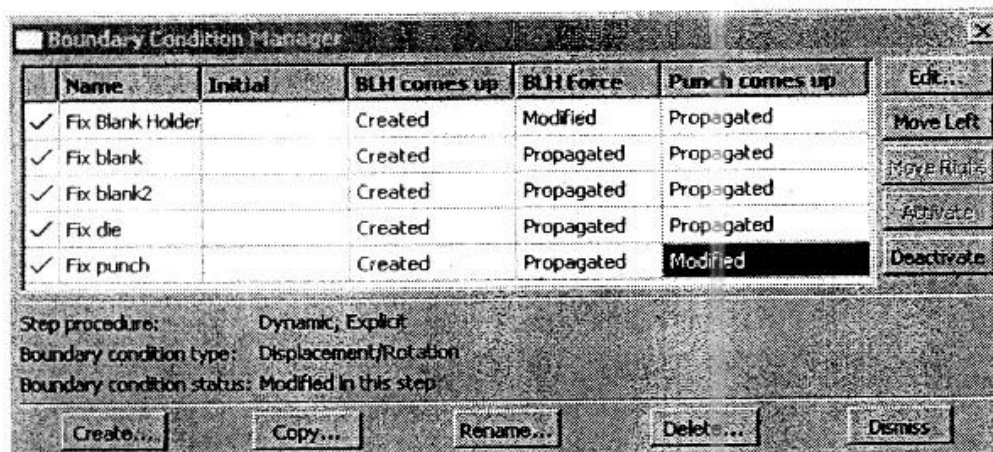
Remove as restrições para mover o blank holder para cima. O blank holder deve ser liberado totalmente no movimento na direção y. Para fazer isto desmarque a caixa u2 usando o boundary condition manager. Ele irá aparecer como modificado.

BC → manager → BLH force → Edit → desmarca a caixa u2.

BC's para o step 3:

Move the punch up to forming operation.

Using boundary condition manager → Edit punch specifying 70 for u2 (vai andar 70 para cima - positivamente) → select amplitude punch.



Fazer o **carregamento do prensa chapas**:

Load → create → name: Blank holder force → (step) BLH force → mechanical → concentrated force → continue → RP do holder → done → CF1 = 0; CF2 = 10000; CF3 = 0 → amplitude → select BLH (não instantaneous) → ok (deve estar propagated for step punch comes up – load manager). Deve aparecer o carregamento do holder no RP – seta para cima.

MESH

Primeiro selecione somente a chapa.

Seleciona part → duplo click chapa → mesh → mesh controls → quad → free → ok

Mesh → element type → explicit → shell → verificar se está com reduced integration (S4R) → Ok.

Seed → part → size → 4.0 → Ok → done
 mesh → part → Yes

JOB

Job → create → name: _____ → continue → descrição → Ok
 Job → submit → name

Job → monitor → name

POSTPROCESSING

VERIFICATION

File → open → file.odb → tools → display group → create → part instance → sheet → replace → dismiss → click deformed shape options in the prompt area → animate: time history

Visualizar deformação plástica ou tensões (Misses):

Results (menu principal) – Field outputs – PE – máxima ou mínima (escolher).

Visualizar → Tools → query → probe values → ok → select variables → passa o mouse pela chapa deformada e pode visualizar os dados para cada nodo localizado.

Obs.: the file abaqus.rpt (gerado no temp) contém os dados para plotar em excel (pode abrir com o wordpad).

Grava os dados como nome.rpt → it will be saved at temp directory.

To plot in excel:

Go to results → History output and select what you want (selected variables at file.cae history output).

To plot punch force (dislocation by force):

Select → reaction force: RF2 PI: punch – 1 → save as → name → punch force

Or

Punch displacement (dislocation by time):

Select → Spatial displacement: U2 PI: punch – 1 – save as → name → punch displacement → dismiss

Report → XY data → chose punch displacement → ok

Report → XY data → chose punch force → ok → create abaqus.rpt file at temp directory

Open excel → file → open → all files → abaqus.rpt → ok → delimited → next → select by tab and space → next → general → finish → multiply punch force by 4 (to compare with experimental) = (C45*4) → Abaqus gives force at N (divide by 1000 to get kN) → select data → plot → standard types → XY (scatter) → smoothed lines → finish.

To plot thickness by distance:

Tool → path → create → name: thickness → node list → continue → blank-1 → add before → select nodes → done → ok

Tools → XY data → create → path → continue → select the path → true distance → select the step/frame → Field output → select STH → ok → ok → save as → name: thickness → ok → plot (in abaqus).

Or

Report → XY → thickness → ok (create abaqus.rpt file at temp directory)

Open excel → file → open → all files → abaqus.rpt → ok → delimited → next → select by tab and space → next → general → finish → select data → plot → standard types → XY (scatter) → smoothed lines → finish.

To create a node list path:

1. Click the tool to display a contour plot of the model. Use the contour options to display the node labels.
2. From the main menu bar, select Tools → Path → Create. The create path dialog box appears.
3. Name the path displacement. Accept the default selection of Node list as the path type, and click Continue. The edit Node List Path dialog box appears.
4. In the path definition table, PART – 1-1 field, and for example, 1:601:40 in the node labels field and press [ENTER]. (This input specifies a range of nodes from 1 to 601 at increments of 40). Alternatively add before...or add after...in the edit node list path dialog box. The selected path highlighted in the current viewport.
5. Click Ok to create the path and to close the edit node list path dialog box.

To save animations:

From the main menu bar, select **Animate** → **Save As**.

The **Save Image Animation** dialog box appears.

1. Use the **Selection** field to choose the viewports to save:
 - a. Click the arrow next to the **Capture** field to save **All Viewports** (the default) or the **Current Viewport**.
 - b. Toggle **Capture viewport decorations** to control whether ABAQUS/CAE saves visible viewport decorations.
2. Use the **Settings** field to specify the name and format of your animation file:
 - a. Click **Select** to filter and browse file names, or enter the file name of your choice in the **File name** field.
3. Click the **Format** button to choose either AVI (the default) or QuickTime format for your file. To learn more about these format options,
4. Click **Apply** to save your animations.
ABAQUS/CAE captures canvas viewports to a file according to your specifications.

ABAQUS/Explicit units:

Units:N,mm

Length=mm

Force=N

Mass=Tonne(1.0E3kg)

Time=second

stress=MPa(N/mm²)

Energy=J

Density=Tonne/mm³

Conductivity=J/(s.mm.C)

Specific Heat = J/(Tonne.C)

????? To display an X-Y plot:

1 – From the main menu bar, select RESULT → HISTORY OUTPUT. ABAQUS displays the HISTORY OUTPUT dialog box.

2 – Click the VARIABLES tab, if it is not already selected. To see the complete description of the variables choices, increase the width of the HISTORY OUTPUT dialog box by dragging the right or left edge.

3 – The OUTPUT VARIABLES field contains a list of all the variables in the history portion of the output database. Select the vertical motion of rigid body reference node Spatial displacement: U2 at node 1000 in **Nset punch** if it is not already selected.

4 – The HISTORY OUTPUT dialog box allows you to select where in the history data the X-Y plot should begin and end; in the most cases the X-axis is assumed to be time. To create as X-Y plot using data in all three steps, do the following:

- a. Click the **Steps/Frames** tab
- b. Drag the cursor over all three steps, if they are not already selected.

You can also choose the frequency at which to read the frames. For the tutorial you can accept the default setting of **Frames: Read all**.

5 – From the buttons across the bottom of the HISTORY OUTPUT dialog box, click plot.

Creating the stress versus time strain versus time data objects

To create the X-Y data objects:

1 – From the main menu bar, select tools → XY data → manager
The XY data manager, click create.

2 – From the XY data manager, click create.

3 – From the create XY data dialog box that appears, select ODB history output if it is not already selected and click Continue. The history output dialog box appears.

4 – In the history output dialog box, do the following:

- a. Click the variables tab.
- b. In the output variables field, select Logarithmic strain components:
- c. Click the Steps/Frames tab.
- d. Select Step 1.
- e. Click Save as. The Save XY data as dialog box appears.
- f. Name the X-Y data strain, and click OK.

5 – Now you are ready to continue the two data objects to create a stress versus strain data object.

6 – Dismiss the history output dialog box.

