MANUAL
SUPERPLASTIC FORMING SIMULATION WITH ABAQUS 6.4-1

Sample input file *.inp	
Section	**Commands**
HEADING	*HEADING step 1 static step 2 expand SUPERPLASTIC FORMING OF cylinder WITH MEMBRANES - auto LOADING 4/14/07 DA Wang: started HT Pham: modified (c) 2007, Dung-An Wang, all rights reserved Products: ABAQUS/Standard This program simulates the SPF of a tube using an axisymmetric model
PART DEFINITION *(1st method)*	*NODE 1, 0., 45.3891 50, 16., 29.3891 100, 0., 13.3891 500, 0., 29.3891 1000, 0., 25.2448 *NGEN, LINE=C, SYSTEM=RC 1, 50, 1, 500 50, 100, 1, 500 *ELEMENT, TYPE=Max1 1,1,2 *ELGEN, ELSET=elset_part 1,99,1,1 *NSET,ELSET=elset_part, NSET=nset_part *nset, nset = nset_monitor 100 *MEMBRANE SECTION, ELSET=elset_part, MATERIAL=mat_SUPRAL 2,
(2nd method)	**include, input=<FILE.inut>* **MEMBRANE SECTION, ELSET=elset_part, MATERIAL=mat_SUPRAL* *2.,* **RIGID BODY, ANALYTICAL SURFACE=elset_die, REFNODE=nset_center*
MATERIAL DEFINITION	*MATERIAL, NAME=mat_SUPRAL *ELASTIC 71000, .34 *CREEP, LAW=TIME 3.114E-5, 2., 0.
DIE DEFINITION *(1st method)*	*SURFACE, TYPE=SEGMENTS, NAME=surf_die START, -6.1883, 2.1448 CIRCL, 6.1883, 2.1448, 0., 10. CIRCL, 23.5399, 35.0093, -22.154, 38.1215 CIRCL, 22.8882, 36.6266, 21.5445, 35.1452 CIRCL, 22.5765, 39.7937, 24.4807, 38.3822 CIRCL, 22.5694, 42.1853, 20.9699, 40.9847 CIRCL, -6.1883, 52.7778, 0., 25.2448

	*RIGID BODY, ANALYTICAL SURFACE=surf_die, REFNODE=1000
CONTACT DEFINITION	*SURFACE, NAME=surf_part, Type=ELEMENT elset_part,SPOS *CONTACT PAIR,INTERACTION=contact_part, smooth=0.2 surf_part, surf_die *SURFACE INTERACTION, NAME=contact_part ***FRICTION **0.1
BOUNDARY AND INITIAL CONDITIONS	*BOUNDARY 1, 1 100, 1 50, 2 1000,1,6 *INITIAL CONDITIONS, TYPE=STRESS, UNBALANCED STRESS=STEP elset_part, 6.89e-3, 6.89e-3 *AMPLITUDE,DEFINITION=SOLUTION DEPENDENT,NAME=AUTO 1.,0.1,1000.
STEP 1	
STATIC SIMULATION	*STEP, INC=100000, NLGEOM, unsymm=yes *STATIC 2.E-3,1.0,1.e-5, 1e-1 *DLOAD elset_part,P, 1e-3
OUTPUT	*monitor, dof=2, node= nset_monitor, frequency=1 *CONTACT PRINT, FREQUENCY=1 *CONTACT FILE, FREQUENCY=1, NSET=nset_part *EL PRINT, ELSET=elset_part , FREQUENCY=1 S, E CE, SINV, *PRINT, CONTACT=YES *NODE FILE, NSET=nset_part, FREQUENCY=1 U, *OUTPUT,FIELD, Frequency=1 *NODE OUTPUT U,RF *Element Output S, E *END STEP
STEP 2	
VISCO SIMULATION	*STEP, INC=100000, NLGEOM, unsymm=yes *VISCO, CETOL=0.005 2e-3, 935, 2e-5, *DLOAD,AMPLITUDE=AUTO elset_part,P, 3e-1 *CREEP STRAIN RATE CONTROL, ELSET=elset_part, AMPLITUDE=AUTO 1e-3,
OUTPUT	*monitor, dof=2, node= nset_monitor, frequency=1 *OUTPUT,FIELD, Frequency=1

	*NODE OUTPUT U,RF *Element Output S, E *END STEP

DESCRIPTIONS

1. HEADING

– This section is used to define a title for the analysis
– The heading can be several lines long, but only the first 80 characters of the first line will be saved and printed as a heading

2. PART DEFINITION

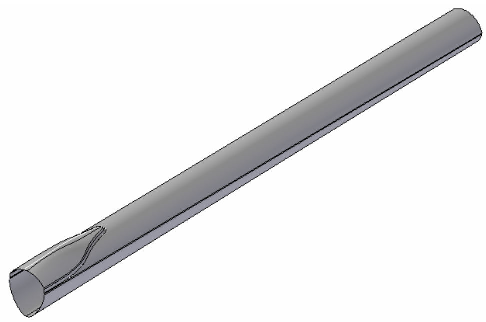

Figure 1. A sample of a final product

– An arbitrary cross section of the product can be made to define the die shape.
– An initial part usually has a simple shape, so we can define it directly in the script file. In this example it is a cylinder of diameter ϕ=32mm

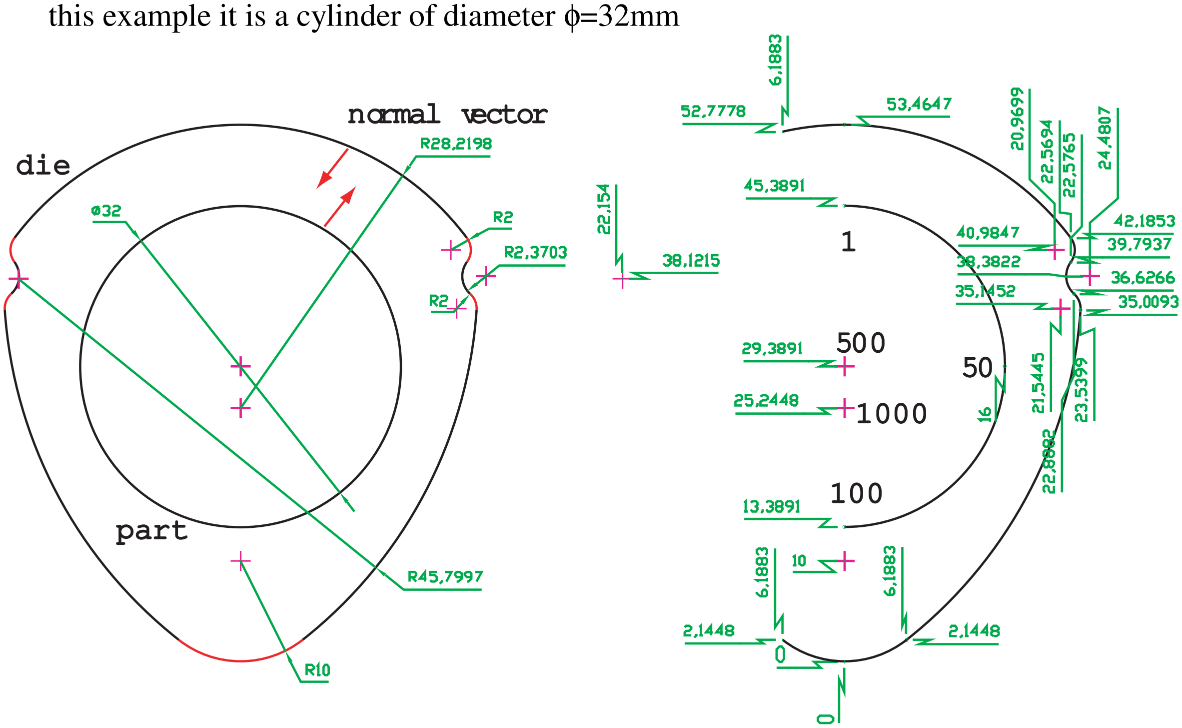

Figure 2. Coordinates of each curve end points.

- To save the computer resources, only half of the model is used for the simulation.
- Refer to [2] for the definition of *NGEN, *ELEMENT, *ELGEN in defining the part
- Read the regular axisymmetric membrane element type MAX1 in [1] for its active DOF (degrees of freedom)

3. MATERIAL DEFINITION

- The Young modulus, poision's ratio and the part material constants A,m,n will be listed here.

4. DIE DEFINITION

- In our simple cross section of the die, it consists of arcs that we can define the whole die by defining coordinates of individual arc (2 end points and its center).
- Note that die nodes are defined CCW in order to have a normal vector pointing toward the part, while the part nodes are defined CW. Two normal vectors of the die and part must point to each other.
- Description in defining SEGMENTS for *SURFACE can be read in [2]
- In this kind of simulation, we assume that the die is a rigid body that isn't affected in any cases from the part. This assumption is somewhat reasonable because the die is usually much larger than the final product. So we do not need to define the material for the die.
- However if we want to know the stress distribution on the die that affect its life, it is still capable to integrate it to the simulation.
- If the cross section is rather complicated or a 3D simulation is required, 2 sections PART DEFINITION and DIE DEFINITION can be replaced by a file.inut, this file can be created by using a meshing software like HyperMesh or any others that one experts

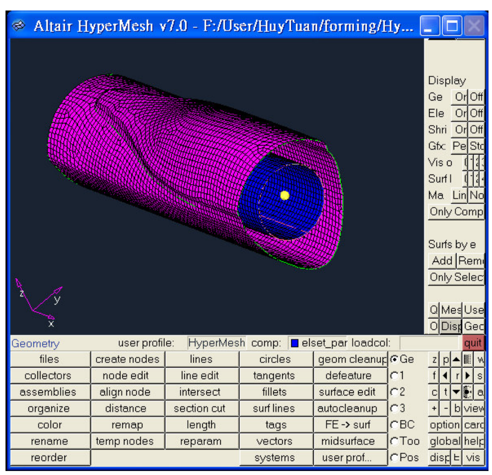

Figure 3. Altair HyperMesh window

- A 3D mesh for part and die of this simulation using HyperMesh may look like as figure 4

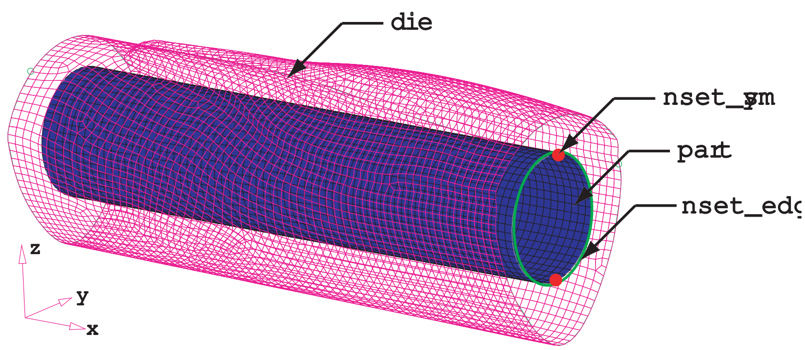

Figure 4. 3D schematic view

Sample of a file.inut	
Heading	** ** ABAQUS Input Deck Generated by HyperMesh Version : 7.0 ** Generated using HyperMesh-Abaqus Template Version : 7.0 ** ** Template: ABAQUS/STANDARD 3D **
Node definition	*NODE 69814, 560.0 , 0.0 , -2.5 . . .
Part definition	*ELEMENT,TYPE=M3D4,ELSET=elset_part 8982, 79967, 79966, 79989, 79985 . . .
Die definition	*ELEMENT,TYPE=R3D4,ELSET=elset_die 4028, 74286, 74748, 74855, 74287 . . .
Boundary condition node sets	*NSET, NSET=nset_edge2 79364, . . . *NSET, NSET=nset_sym 79248, . . . *NSET, NSET=nset_edge1 79248, . . . *NSET, NSET=nset_monitor 79362, *NSET, NSET=nset_center 69814, *NSET, NSET=nset_part 79248, . . .

5. CONTACT DEFINITION

- This section defines the contact between the part and die. We must define 2 contact surfaces, the die is a master surface and the part is a slave surface.
- Friction can also be added to the simulation to make it get along well with practice

6. BOUNDARY AND INITIAL CONDITIONS

– Practical boundary conditions will give good results.

7. STEP 1

– The initial application of the pressures is assumed to occur so quickly that it involves purely elastic response. This is achieved by using the *STATIC procedure

8. STEP 2

– The creep response is developed in a second step using the *VISCO procedure

9. RUN THE SIMULATION

– In the MS-DOS screen, change directory to the input file <filename.inp>, from the command prompt type

D:\..\input file directory> abaqus job=<filename> interactive

10. POSTPROCESSING

– We will use the ABAQUS/CAE to access the results of simulation
– Sample result for a 2D simulation

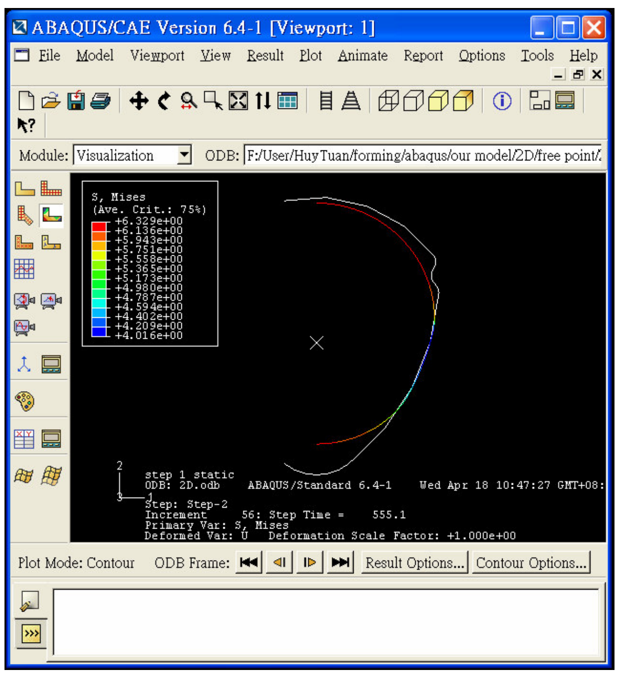

Figure 5. ABAQUS/CAE window

◆ 3 fix nodes: (node 1, 100 dof 1 is zero, node 50 dof 2 is zero)

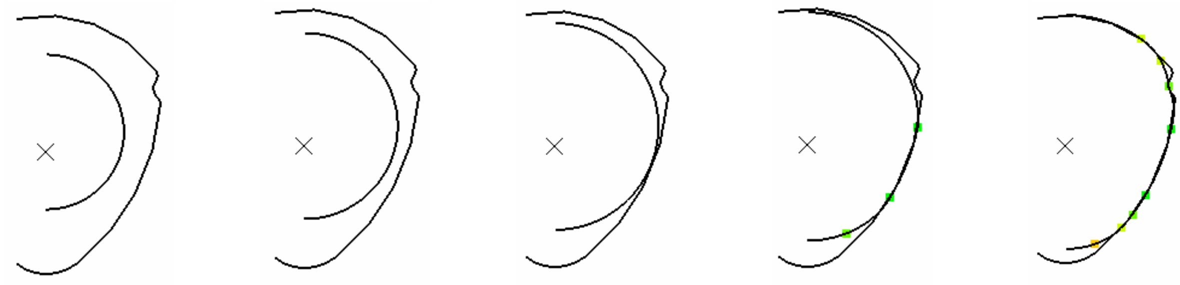

Figure 6. The deformation history

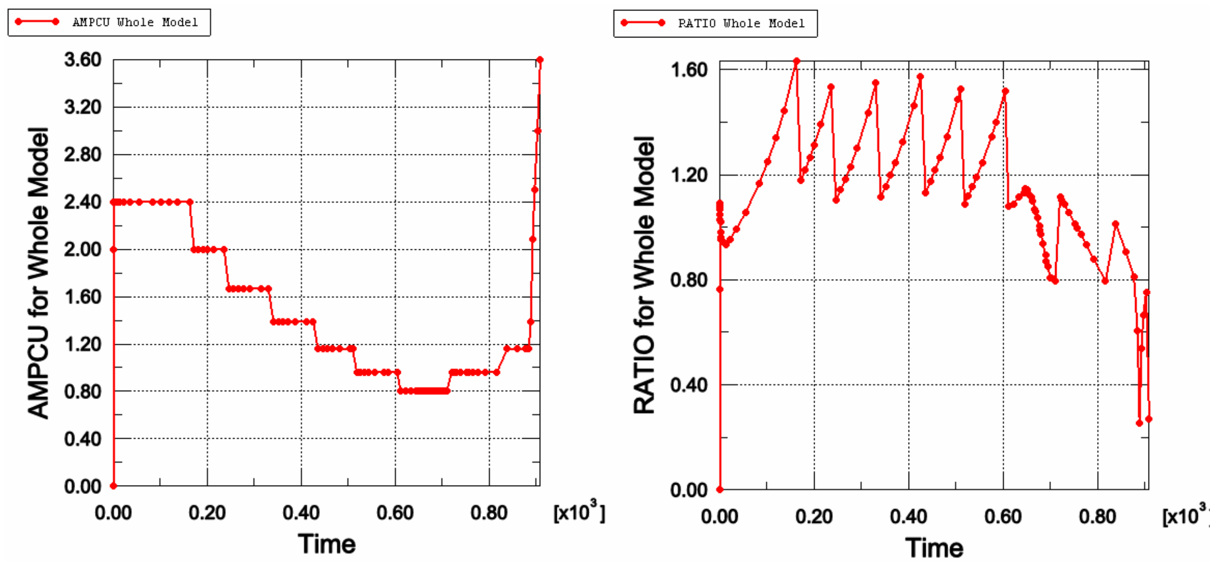

Figure 7. Data history

(a). Pressure cycle,

(b). History of ratio between maximum creep strain rate and target creep strain rate

– Sample results for a 3D simulation

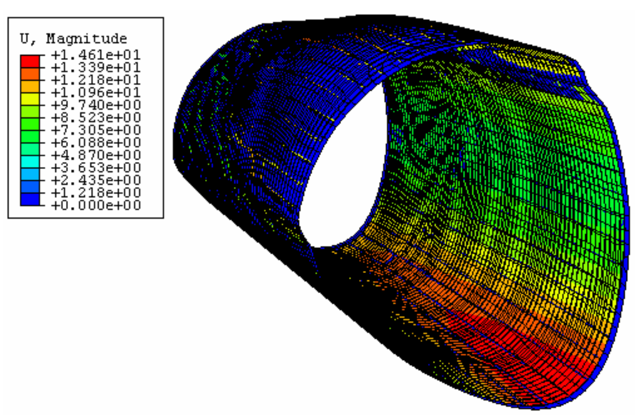

Figure 8. A 3D simulation

REFERENCES

[1]. ABAQUS Analysis User's Manual

[2]. ABASQUS keyword reference Manual