52 Hydro-forming of a Square Pan

- Summary 1059
- Introduction 1060
- Modeling Details 1060
- Solution Procedure 1063
- Results 1064
- Modeling Tips 1065
- Input File(s) 1066
- Video 1066
# Summary

<table>
<thead>
<tr>
<th>Title</th>
<th>Chapter 52: Hydro-forming of a Square Pan</th>
</tr>
</thead>
<tbody>
<tr>
<td>Features</td>
<td>Large Deformation Hydro-Forming of Panels under Pressure loading</td>
</tr>
</tbody>
</table>

## Geometry

- **Body 1: hex8**
- **Body 2: quad4**

### Material properties

- $E = 69 GPa$, $\nu = 0.3$, No hardening $Y = 240 MPa$

### Analysis characteristics

- Quasi-static contact analysis using adaptive time stepping with artificial damping, geometric, and material nonlinearity due to large strain.

### Boundary conditions

- Only one quarter of the plates are modeled due to symmetry. Symmetric conditions are applied along with other suitable boundary conditions to suppress rigid body motions.

### Applied loads

- Pressure applied on the upper surfaces of the solid and shell elements, respectively.

### Element types

- CHEXA and CQUAD4 with assumed strain formulation

### FE results

- Displacement contours on formed shape
Introduction

This example demonstrates the ability of the MD Nastran SOL 400 nonlinear solution sequence to hydro-form a square pan using pressure loads. Such problems exhibit geometrical instability because of the formation of wrinkles during the forming process, posing substantial challenges for the analysis. These challenges require a robust algorithm to steer through convergence process during the forming. Artificial damping, demonstrated herein, shows its capability to deal with such wrinkling instabilities.

Modeling Details

A numerical solution has been obtained with MD Nastran's SOL 400 for a 3-D representation of the structure with deformable-to-deformable contact between the two panels. The details of finite element model, contact simulation, material, load, boundary conditions, and solution procedure are discussed below.

The case control section of the input contains the following options for nonlinear analysis:

```
SOL 400
CEND
SUBCASE 1
STEP 1
  TITLE=This is a default subcase.
  ANALYSIS = NLSTATIC
  NLSTEP = 1
  BCONTACT = 1
  SPC = 2
  LOAD = 2
  DISPLACEMENT (SORT1, PLOT, REAL) = ALL
  NLSTRESS (SORT1, PLOT, REAL) = ALL
  BOUTPUT (SORT1, PLOT, REAL) = ALL
```

The input data shown above defines the analysis type and loading sequences. **ANALYSIS = NLSTATIC** means that the type of this SOL 400 analysis step is nonlinear quasi-static. One load step defined in the subcase. This step comprises the pressure load application onto the surfaces of the solid and shell elements. The control parameters for the load stepping and iterative procedures are defined by the bulk data option NLSTEP. The contact table and contact parameters are given via BCONTACT. The displacement or rotational constraints are applied via SPC1. The pressure loads are applied via two PLOAD4 entries, one for the shell surface (body2) and the other for the upper surface of the solid body (body1).

The large strain and geometrical nonlinear characteristics are activated by the NLMOPTS/LRGS and LGDISP options, respectively:

```
NLMOPTS  LRGS   1
PARAM    LGDISP  1
```

where the NLMOPTS entry field LRGS = 1 triggers the large strain formulation and provides better behavior when plasticity is present. LGDISP = 1 indicates the use of large displacement, large rotation kinematics for the element.

The boundary conditions and pressure loads applied onto the two panels are shown in Figure 52-1. Due to symmetry, only one quarter of the structure is shown.
Element Modeling

Besides the standard options to define the element connectivity and grid coordinate location, the bulk data section contains various options which are especially important for nonlinear analysis. The nonlinear extensions to the lower-order solid element, CHEXA can be activated by using the PSLDN1 property option in addition to the regular PSOLID property option:

```
PSOLID    1       1       0
PSLDN1   1       1
C8      SOLID   LRIH
```

The PSLDN1 option allows the element to be used in both large displacement and large strain analysis and has no restrictions on the kinematics of deformation unlike the regular CHEXA elements with only the PSOLID property entry. The 8-node hexahedral element with the reduced integration scheme is very suitable for problem with severe bending deformation.

The other panel is modeled with shell elements, and activated via two bulk data cards: PSHELL and PSHLN1. PSHELL is the entry to define the regular shell element in Nastran. PSHLN1 is an extra entry for user to select the advanced shell element which is more suitable for large strain and rotations:

```
PSHELL   2       1      2.54     1               1
PSHLN1   2
```

where 2.54 mm is the thickness of the shell elements.

Modeling Contact

In this example, contact occurs between the solid and shell panels between which glued contact is defined. Therefore no friction is considered.
To identify how the contact bodies can touch each other, the BCTABLE option is used. BCTABLE with ID 0 is used to define the touching conditions at the start of the analysis. This is a mandatory option required in SOL 400 for contact analysis and it is flagged in the case control section through the optional BCONTACT = 0 entry. The BCTABLE with ID 1 is used to define touching conditions for later increments in the analysis, and is flagged using BCONTACT = 1 in the case control section. Also, the SLAVE-MASTER combination specifies that the corresponding body is a slave or master body. This, in literature, is variously referred to as either contacting body nodes or tied nodes (imagining the situation of multi-point constraints). The nodes belonging to body 1 are said to belong to the master body, similarly referred to as the contacted body nodes or the retained nodes.

```
BCTABLE  0                       1
  SLAVE   2   0.    0.    0.    0.    3
        1   0    0
      FBSH 1.+20  .9   0.
  MASTERS 1
BCTABLE  1                       1
  SLAVE   2   0.    0.    0.    0.    3
        1   0    0
      FBSH 1.+20  .9   0.
  MASTERS 1
```

The BCBODY entry defines the deformable body including the body ID, dimensionality, type of body, type of contact constraints and friction, while the BSURF entry identifies the elements forming each part of the deformable bodies:

```
$ Contact Body: DeformBody_SHELL  
BCBODY         1      3D  DEFORM       2                       0  
BSURF          2       1    THRU     525  

$ Contact Body: DeformBody_SOLID  
BCBODY         2      3D  DEFORM       3                       0  
BSURF          3     526    THRU     925
```

### Material Modeling

The isotropic, Hookean elastic material properties of the deformable body are defined using the following MAT1 option as follows:

```
MATEP    1       Perfect240.                     Isotrop Addmean  
MAT1     1      69000.          .3  
```

The Young's modulus is taken to be 69 GPa with a Poisson's ratio of 0.3. Perfect plasticity is assumed with yield at a stress of 240 MPa.

### Loading and Boundary Conditions

Symmetry conditions are applied to the nodes along the X-axis and Y-axis. To remove rigid body motion, the boundary nodes of the shell panel are constrained in the Z-axis.

```
SXNAME LOADCOL 3       SPC1    "CONSTRAIN-UZ"
SPC1           3       3       1    THRU      26
SPC1           3       3      27      53      79     105     131     157+
```
CHAPTER 52
Hydro-forming of a Square Pan

+ 183 209 235 261 287 313 339 365+

$SXNAME LOADCOL 4 SPC1 "SYMM-SHELL-UX"
SPC1  4  156  26  52  78 104 130 156+
+ 182 208 234 260 286 312 338 364+

$SXNAME LOADCOL 5 SPC1 "SYMM-SHELL-UY"
SPC1  5  246  561 THRU 576

$SXNAME LOADCOL 6 SPC1 "SYMM-SOLID-UX"
SPC1  6  1 1455 THRU 1458

$SXNAME LOADCOL 7 SPC1 "SYMM-SOLID-UY"
SPC1  7  2 1377 1378 1381 1382 1385 1386+
+ 1389 1390 1393 1394 1397 1398 1401 1402+

SPCADD 12 3 4 5 6 7

The loading involves the application of pressure on the upper surfaces of the two bodies:

$SXNAME LOADCOL 1 PLOAD4 "PRESSURE-SHELL"
PLOAD4  1  1 -0.4
PLOAD4  1  2 -0.4
PLOAD4  1  3 -0.4

$SXNAME LOADCOL 2 PLOAD4 "PRESSURE-SOLID"
PLOAD4  2  526 0.4 578 586
PLOAD4  2  527 0.4 580 588
PLOAD4  2  528 0.4 584 592

LOAD 13 1. 1. 1 1. 2

Load ID 1 represents the distributed load applied to the solid panel. Load ID 3 is the pressure applied on the shell panel. As shown above, the pressure applied is 0.4 MPa.

Solution Procedure

Adaptive time-stepping defined through the NLSTEP card is used in the example. Two flavors of adaptive stepping are demonstrated: (a) a scheme using artificial damping in nug_52a and (b) a scheme without damping in nug_52b.

The step sequence control card (NLSTEP) in nug_52a is shown below:

NLSTEP  1  1. +
+ GENERAL 25 10 +
+ ADAPT 1.00E-2 1.E-5 0.50 1.2 -1 999999 +
+ MECH UPV 0.1 0.1 PFNT -1

Salient parameters defined through the above cards are as follows:

Total Time (2nd field on 1st card): 1.0
Max. Number of recycls (1st field on GENERAL card): 25
Initial time step (1st field of 1st line of ADAPT card): 0.01
Desired Number of Recycles (4th field of 1st line of ADAPT card): blank - defaults to 4
Scale Factor (5th field of 1st line of ADAPT card): 1.2
Output Control (6th field of 1st line of ADAPT card): -1 (only last increment is output)
Damping flag (1st field of 2nd line of ADAPT card): 4
Convergence control flag (1st field of MECH card): UPV (vector components of displacements and residuals)
Convergence tolerances (2nd and 3rd fields of MECH card): 0.1

The parameters in nug_52b are identical with the exception of the damping flag - it is set to 0.

The damping algorithm in nug_52a only uses the initial time step information of the ADAPT card. It does not use the desired number of recycles or the scale factor. The time stepping algorithm (increase of 1.5 or cutback of 0.5) is based on a comparison of the incremental strain energy and the estimated damping energy.

The recycle based algorithm in nug_52b is based on a comparison of the actual number of recycles taken for convergence to the desired number of recycles. The time step is allowed to increase if the number of actual Newton-Raphson recycles < the desired number.

Results

Figure 52-2 shows the sequence of analysis with a close-up view of the square pan when damping is used for the analysis. It can be seen that a wrinkle develops with the increase of applied pressure. For comparison purposes, another job which does not use damping is also shown in Figure 52-3. It can be seen that the wrinkles develop more slowly with loading. Without damping applied, the wrinkling occurs a little earlier and the analysis is extremely unstable. That is why significantly more iterations are needed at the wrinkling stage. In some cases, this may cause an unsuccessful analysis if the control parameters are not properly set.
Modeling Tips

Artificial damping typically acts as an extra strategy for analyses involving wrinkling and snap-through (softening of materials) for both uniform and adaptive stepping strategies. Furthermore, for the recycle based approach, proper setup of the adaptive time stepping control parameters can help when conducting nonlinear analysis. In general, the most significant parameters that influence the number of increments / number of iterations are the initial time step and the desired number of recycles. Some broad guidelines are as follows: For linear problems, the initial time step ratio can be set as 1.0 and the desired number of recycles can be set as 4. For mildly nonlinear problems, the initial time step
ratio can be set as 0.1 and the desired number of recycles can be set between 6 - 8. For highly nonlinear problems, the initial time step ratio can be set as 0.01 and the desired number of recycles can be set between 8 - 10.

Contact analyses, especially with friction, often demonstrate strong nonlinearities due to the frequent touching of a node or the sliding of a node from its contacting surface. Adding artificial damping will also help improve the stability and the efficiency of such contact analyses. It is to be noted that in a glued contact between solid and shell elements, the contact body comprising of the shell elements are to be modeled as the slave.

The default values for the adaptive time stepping scheme in Sol 400 represent a serious attempt to ensure successful analysis completion in the general case. However, each problem will be helped by an intelligent choice of control parameters. Generally, to start with a small time step is always better than a large initial time step, even though the cut back will reduce the time step, it will cost more time completing the analysis. Also, it is recommended adjusting the desired number of iterations accordingly if the convergence criteria change. For example, a tight convergence tolerance requires more iterations than a loose tolerance. In the case of contact, a higher number of desired iterations can speed up the analysis without scarifying accuracy.

For comparison purposes, both input decks are included with this demonstration.

### Input File(s)

<table>
<thead>
<tr>
<th>File</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>nug_52a.dat</td>
<td>MD Nastran input for adaptive time stepping with artificial damping</td>
</tr>
<tr>
<td>nug_52b.dat</td>
<td>MD Nastran input for adaptive time stepping without artificial damping</td>
</tr>
</tbody>
</table>

### Video

Click on the image or caption below to view a streaming video of this problem; it lasts approximately 26 minutes and explains how the steps are performed.

![Video of the Above Steps](image)