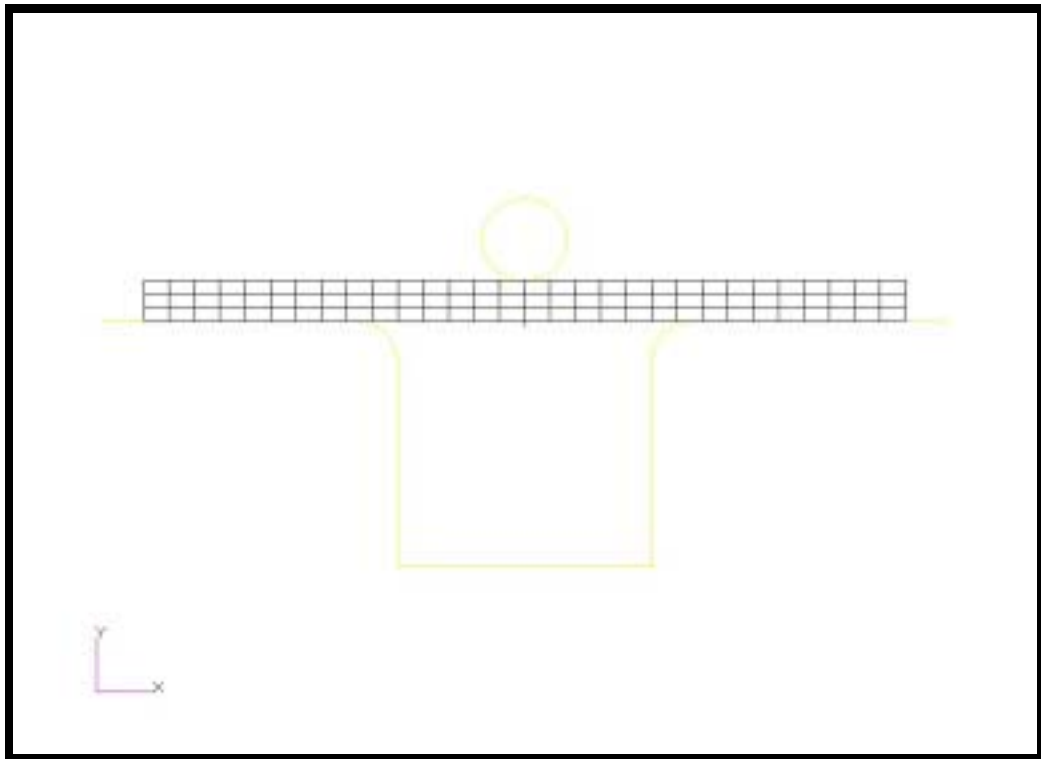


---

## Lesson 8

# *Break Forming*



### **Objectives:**

- Large displacement analysis.
- Contact analysis.
- Plasticity theory.



**Model Description:**

A flat sheet is formed into an angled bracket by punching it through a hole in a table using the contact option.

The cylindrical punch drives the sheet down into the hole of the table to a total stroke of 0.3". The material is elastic plastic with work hardening. At the bottom of the stroke the total plastic strain is nearly 45%. The vertical punch force rises quickly, hardens through about half of the stroke, then softens near the end of the stroke.

This workshop problem exemplifies how every point in the sheet must follow the material's constitutive behavior as well as being in equilibrium throughout the deformation.

---

## **Suggested Exercise Steps:**

- Import the geometry.
- Model the Rigid and Deformable Contact Curves and Surfaces.
- Define the Materials and Properties.
- Define the Load sets.
- Run the analysis.
- Evaluate the results.
- Run the analysis as a DDM.

## Exercise Procedure:

1. Open a new database called **cup.db**.

**File/New ...**

*Database Name:*

**break\_forming.db**

**OK**

Change the *Analysis Preference* to **MSC.Marc**.

*Analysis Code:*

**MSC.Marc**

**OK**

2. Import the model geometry

**File/Import ...**

Change the Source to read from an **IGES** file:

*Source:*

**IGES**

*File Name:*

**s4.igs**

**Apply**

Close the *IGES Import Summary* form that appears by clicking **OK**.

**OK**

3. Now apply the **Load and Boundary Conditions**.

First, fix the plate in the X-direction.

■ **Load/BCs**

*Action:*

**Create**

*Object:*

**Displacement**

*Type:*

**Nodal**

*New Set Name:*

**fixed\_in\_x**

**Input Data...**

*Translations:*

< 0, , >

**OK**

**Select Application Region...**

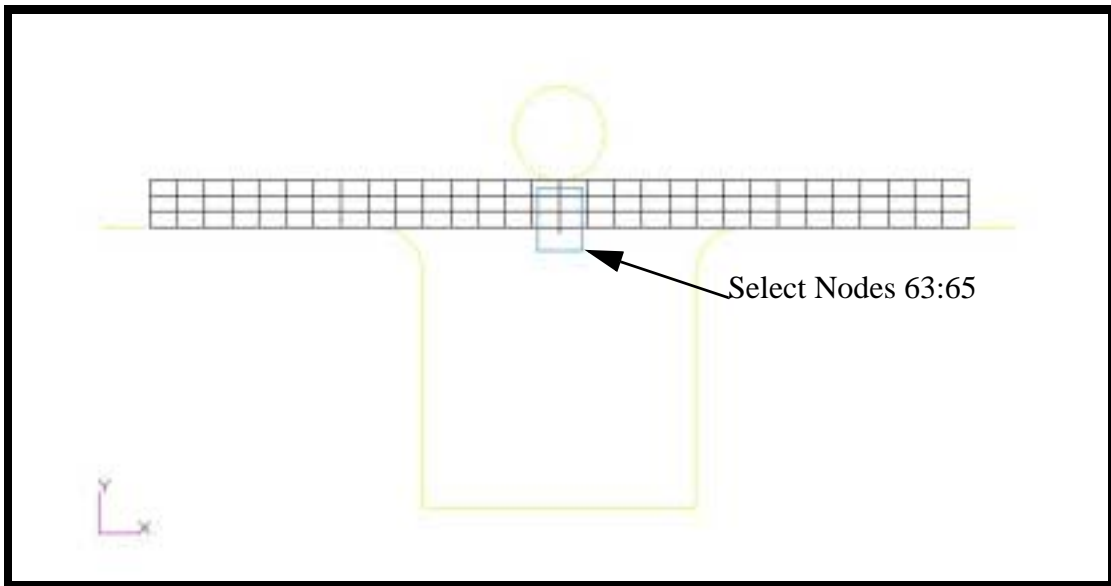
*Geometry Filter:*

● **FEM**

*Select Nodes:*

see Figure 8.1

**Figure 8.1 - Fix selected nodes on the plate in the x-direction**



**Add**

**OK**

**Apply**

4. Now create the **Load and Boundary Conditions** for the contact. First, begin with the plate as the deformable body.

*Action:*

**Create**

*Object:*

**Contact**

*Method:*

**Element Uniform**

*Option:*

**Deformable Body**

*New Set Name:*

**plate**

# Break Forming

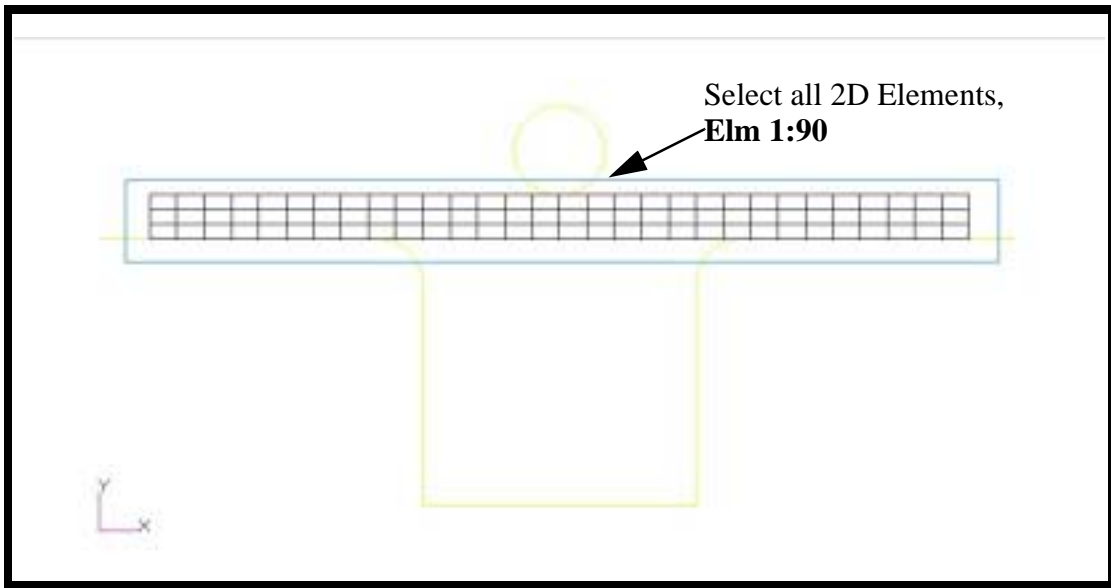
Target Element Type:

**Select Application Region...**

Geometry Filter:  FEM

Select 2D Elements:

**Figure 8.2 - Deformable contact elements**



Next, develop the contact LBCs for the table rigid body.

Action:

Object:

Option:

New Set Name:

Target Element Type:

**Select Application Region...**

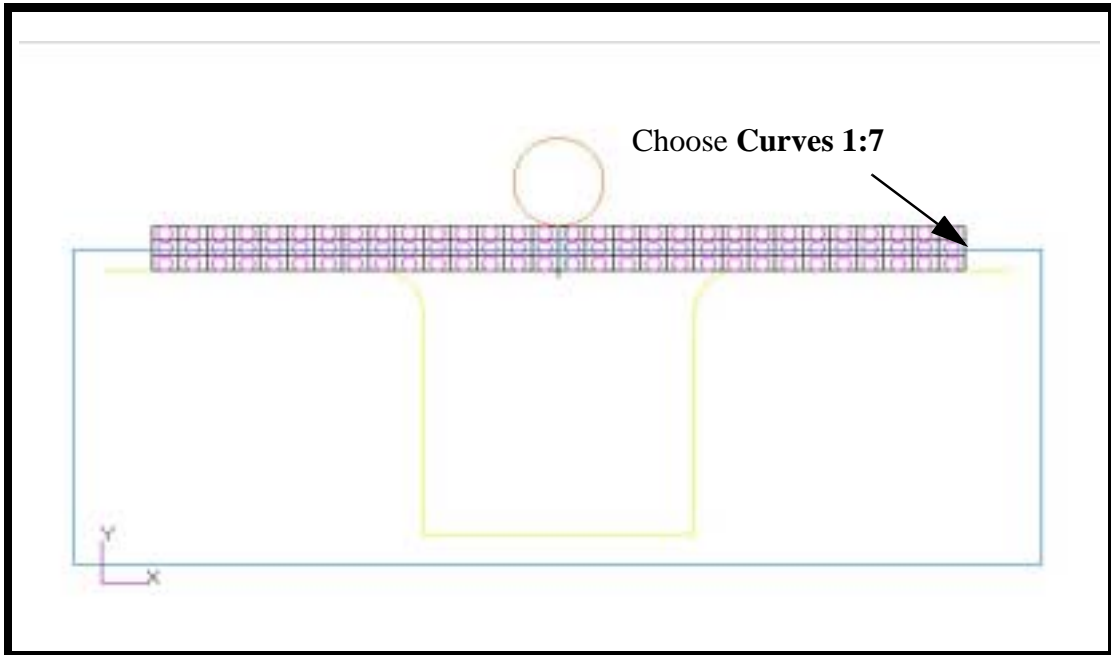
Geometry Filter:

● Geometry

Select Curves:

see Figure 8.3

Figure 8.3 - Rigid contact plate curves



Add

OK

Apply

Finally, create the contact LBCs for the rigid punch.

Action:

Create

Object:

Contact

Option:

Rigid Body

New Set Name:

punch

Target Element Type:

1D

Input Data...

Motion Control:

Velocity

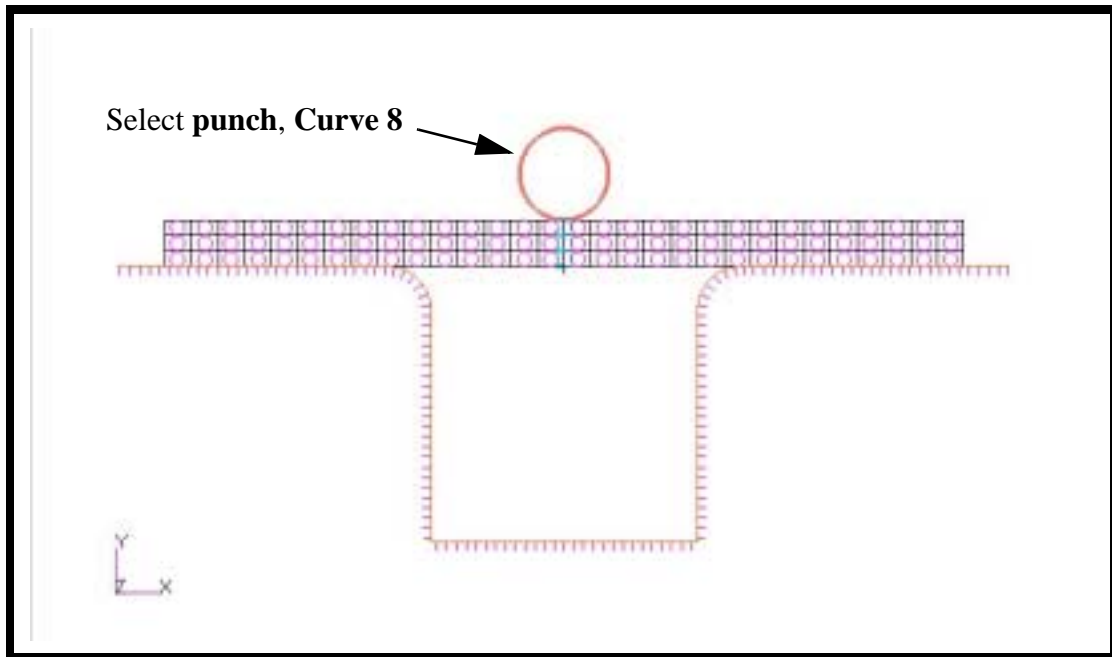
Displacement:

<0, -0.6, 0 >

OK

**Select Application Region...***Geometry Filter:*● **Geometry***Select Curves:*

see Figure 8.4

**Figure 8.4 - Rigid punch contact****Add****OK****Apply**

Note: The tick marks on the **punch** should be pointing inwards.

5. Create the Stress-Plastic Strain Curve needed for Material Properties.

In this step you will create a tabular field which represents an Equivalent Stress versus Plastic Strain response of a certain point in the plate.

**■ Fields***Action:***Create**

---

*Object:* **Material Property**

*Method:* **Tabular Input**

*Field Name:* **plastic\_strain**

*Active Independent Variable:*  **Temperature (T)**  
 **Strain (e)**

**Input Data...**

The *1D Material Scalar Table Data* form needs to be filled out as shown in Table 1. To fill in the table, click on the cell you wish to edit,

**Table 1: Stress vs. Plastic Strain data**

Plastic Strain	Stress
<b>0</b>	<b>5e4</b>
<b>0.1</b>	<b>6.3e4</b>
<b>0.2</b>	<b>6.9e4</b>
<b>0.3</b>	<b>7.4e4</b>
<b>0.5</b>	<b>8.3e4</b>
<b>0.8</b>	<b>9.4e4</b>
<b>1.</b>	<b>1e5</b>

enter the value in the *Input Scalar Data* databox and then press <Return>. The table will automatically tab down.

1D Material Scalar Table Data		
Input Scalar Data <input type="text"/>		
Data		
	e	Data
1	0.00000E+00	5.00000E+004
2	1.00000E-001	6.30000E+004
3	2.00000E-001	6.90000E+004
4	3.00000E-001	7.40000E+004
5	5.00000E-001	8.30000E+004
6	8.00000E-001	9.40000E+004
7	1.00000E+000	1.00000E+005
<input type="button" value="OK"/>		

6. Create the material **steel** for the plate.

In this step you will create one material, **steel**, with both linear elastic and plastic constitutive material properties. The plastic constitutive material property will be created from the stress versus plastic strain curve. First, create the linear elastic constitutive model.

### ■ Materials

Action:

Object:

Method:

Material Name:

---

*Constitutive Model:*

*Elastic Modulus:*

*Poisson's Ratio:*

Define the plastic portion of the curve.

*Constitutive Model:*

*Stress vs. Plastic Strain:*

7. Create the Element Properties for the steel plate.

Here, an Element Property for the plate is created, modeled as an axisymmetric 2D Solid, using the previously created steel material.

### ■ Properties

*Action:*

*Dimension:*

*Type:*

*Property Set Name:*

*Material Name:*

After clicking in the *Select Members* text box, choose the **Shell Element** icon.



**Shell Element**

*Select Members:*

## Break Forming

---




8. Create the Analysis Steps and submit the Analysis.

In this section, you are going to create each of the consecutive steps for the analysis. For each step, you will define the step name, solution type, solution parameters, corresponding load case, and output requests.

### ■ Analysis

*Action:*

*Object:*

*Method:*

*Job Name:*

#### Load Step Creation...

*Job Step Name:*

*Solution Type:*

#### Solution Parameters...

#### Load Increment Parameters...

*Total Time:*

*# of Steps of Output:*






Select only the necessary load step.

#### Load Step Selection...

*Selected Job Steps:*

---

**OK**

**Apply**

At this point the analysis is running. You can monitor the analysis by monitoring the *bracket.sts*, *bracket.log*, and *bracket.out* files. A more convenient method would be to use the **Monitor** action in the **Analysis** application.

■ **Analysis**

*Action:*

**Monitor**

*Object:*

**Job**

**View Status File...**

When the end of this file has the following line, you know that the job has completed successfully: Job ends with exit number : 3004

9. Reading in the Results

■ **Analysis**

*Action:*

**Read Results**

*Object:*

**Result Entities**

*Method:*

**Attach**

**Select Results File...**

*Selected Results File:*

**bracket.t16**

**OK**

**Apply**

10. Evaluate the Results by creating a plot of the last increment.

■ **Results**

*Action:*

**Create**

*Object:*

**Quick Plot**

*Select Results Cases:*

select the last increment, at  
time= 0.5s

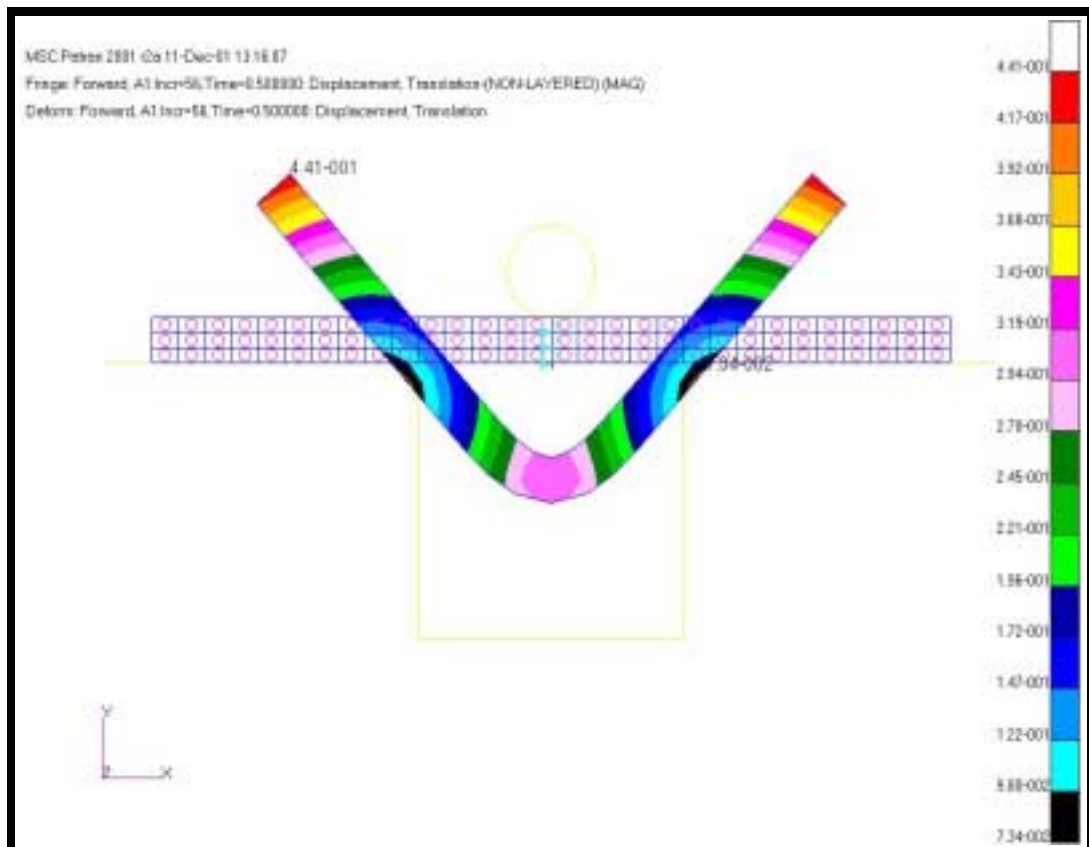
*Select Fringe Cases:*

**Displacement, Translation**

*Select Deformation Result:***Displacement, Translation****Apply**

The results should look like Figure 8.5.

**Figure 8.5 - Last Increment of Punch extruding Plate.**



11. Set up a four-domain DDM job and analyze.

First, you must create the groups that correspond to each domain.

**Group/Create**

*New Group Name*

**D1**

■ **Make Current**

*Groups Contents*

**Add Entity Selection**

---

*Entity Selection*

select the left eight columns of elements,  
**Elm 1:24**

**Apply**

*New Group Name*

**D2**

*Entity Selection*

select the left-center seven columns of elements,  
**Elm 25:45**

**Apply**

*New Group Name*

**D3**

*Entity Selection*

choose the right-center seven columns of elements,  
**Elm 46:66**

**Apply**

*New Group Name*

**D4**

*Entity Selection*

select the right eight columns of elements,  
**Elm 67:90**

**Apply**

**Cancel**

■ **Analysis**

*Action:*

**Analyze**

*Object:*

**Domains**

*Method:*

**Full Run**

**Domain Decomposition...**

*Decompose Domains By:*

**Manually**

*Number of Domains:*

**4**

**Create**

*Domain Information:*

## *Break Forming*

---

*Group:*

for each domain, choose from the *Select a Group* listbox, the corresponding domain group

	<b>Domain</b>	<b>Group</b>
1	Domain 1	<b>D1</b>
2	Domain 2	<b>D2</b>
3	Domain 3	<b>D3</b>
4	Domain 4	<b>D4</b>

**OK**

If run on four different processors, four separate *.dat* files would be created, each being specific to its own respective group. If running the analysis on four processors were available, this should considerably shorten the total cpu time.

When done viewing, close the database and quit PATRAN.

This concludes the exercise.

