Contact Analysis of a Squeezed Disc
## Outline

1 Description
   1.1 Analysis Procedures ................................................................. 4

2 Finite element model
   2.1 Geometry ......................................................................................... 8
   2.2 Properties ......................................................................................... 10
   2.3 Boundary Conditions ...................................................................... 14
      2.3.1 Loads ......................................................................................... 17
      2.3.2 Mesh ......................................................................................... 18

3 Structural Nonlinear Analysis ............................................................. 19
   3.1 Commands ....................................................................................... 19
   3.2 Results .............................................................................................. 26

Appendix A Additional Information ......................................................... 27
1 Description

This tutorial illustrates a two-dimensional contact analysis: a circular disc is squeezed between two rigid plates as shown in Figure 2. The disc $ABCD$ is modeled with continuum plane stress elements and linear elastic material properties. The rigid plates $GH$ and $EF$ are modeled with contact elements and assigned with target contact properties. The edges of the disk are also modeled with contact elements, but with contacter contact properties. An overview of the geometry is shown in Figure 1.

Figure 1: Geometry of the model [m]

Figure 2: Disc between plates
1.1 Analysis Procedures

In DIANA, the solution of nonlinear analyses is achieved using *incremental-iterative solution procedures*, and consist of three factors that must be selected by the user:

- incremental procedure
- iterative solution method
- convergence criteria

The selection of solution procedures that successfully solve a nonlinear analysis is usually based on a *trial-error approach*. The knowledge of the model characteristics plays an important role in the selection of the solution procedures. Examples of these characteristics are:

- load-displacement diagram (identification of equilibrium path of the structure)
- type of external load applied (applied forces, prescribed displacements, loads with time, etc.)
- type of failure (local or global)

Figure 3: Common load vs. displacement paths
Figure 4: Control possibilities

<table>
<thead>
<tr>
<th>Incremental procedure</th>
<th>Increment size input</th>
<th>Iterative solution methods</th>
<th>Convergence criteria</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pure load control</td>
<td>Explicit</td>
<td>Regular Newton-Raphson</td>
<td>Energy norm</td>
<td>- uses prescribed load - increment size is fixed - trace continuous growth behavior - cannot trace softening behavior</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Modified Newton-Raphson</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Constant and linear stiffness</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Secant methods</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Load control with arc-length</td>
<td>Explicit</td>
<td>Idem</td>
<td>Idem</td>
<td>- uses prescribed load - increment size is variable - trace all common load-displacement paths</td>
</tr>
<tr>
<td>Displacement control</td>
<td>Explicit</td>
<td>Idem</td>
<td>Idem</td>
<td>- uses prescribed displacement - increment size is fixed - cannot trace snap-back behaviors - usually for structures with one point load only</td>
</tr>
<tr>
<td>Iteration based adaptive loading</td>
<td>Implicit</td>
<td>Idem</td>
<td>Idem</td>
<td>- uses prescribed load or displacement - increment size is variable - can be combined with arc-length method - trace all common load-displacement paths</td>
</tr>
<tr>
<td>Energy based adaptive loading</td>
<td>Implicit</td>
<td>Idem</td>
<td>Idem</td>
<td>- uses prescribed load - trace all common load-displacement paths - must be combined with arc-length</td>
</tr>
</tbody>
</table>

Pure force control analysis
- Loads (forces, thermal, gravity, etc.) are incrementally applied.
- Pure load control analyses are interested in models with continuous growth behavior.

Force control analysis with Arc-length method
- The pattern of the applied loads is proportionally incremented (using a single load multiplier) to achieve equilibrium under the control of a specified length (arc-length) of the equilibrium path.
- Load control analyses with arc-length method cover all equilibrium paths.

Displacement control analysis
- Prescribed displacement in the direction of a degree of freedom of a reference point on the structure is incrementally applied.
- Displacement control analyses are interested in models with snap-through behaviors.
In this tutorial, to successfully perform the contact analysis, **two** executable blocks are needed:

- **Executable block 1**: to establish the contact process
- **Executable block 2**: to perform the contact process itself

For the contact model, we have:

- possible load vs. displacement path - snap-through or snap-back behavior
- type of external load: prescribed displacement
- failure type expected: local (usual for cracking analysis)

Since the analysis is based on *deform load control*, snap-through load vs. displacement path is assumed as a first attempt.

Thus, the incremental-iterative solution procedure suggested, for a first attempt:

- incremental procedure
  - **Executable block 1**: explicitly load increments
  - **Executable block 2**: cutback based adaptive loading
- iterative solution method: Newton
- convergence criteria:
  - **Executable block 1**: force criterion
  - **Executable block 2**: energy criterion

Keep in mind that a contact problem is highly nonlinear and in its basic form includes contact conditions and geometric nonlinearities.

If the analysis solution becomes difficult to achieve, the application of advanced analysis tools like *Save/Restore Steps* functionality can be very useful, which allows analysis continuation in case of non-convergence or divergence. For more information see the *User’s Manual*. 
2 Finite element model

We start a new project for a structural analysis. We specify a two dimensional project with a model size of 1 km. We set the units to SI units.
2.1 Geometry

We define the geometry of the model. First we create a circle sheet. Subsequently, we create the *contacter* geometry by adding a circle line. Finally, we add two lines for the *target* geometry.
For the *targets* it is important that the local y axes of the mesh is in the direction of the *contacter* mesh. Therefore, *Line 1* and *Line 2* need to be defined with *point 1* and *point 2* as indicated in Figure 12 and Figure 13.

**Main menu** ➔ **Geometry** ➔ **Create** ➔ **Add line** ➔ [Fig. 12]

**Main menu** ➔ **Geometry** ➔ **Create** ➔ **Add line** ➔ [Fig. 13]

**Figure 12**: Add *target* geometry (1)

**Figure 13**: Add *target* geometry (2)

**Figure 14**: Geometry view
2.2 Properties

We assign the material and geometry properties to the shapes. For the circle sheet we use a Young’s modulus of $2.1 \times 10^5$ N/m$^2$, Poisson’s ratio of 0.3 and a mass density of 78 kg/m$^3$. Please note that we also need to specify element data in order to deactivate any EAS for plane stress elements specifying the NOCSHE option.

---

1EAS refer to Enhanced Strain Considerations for lower order continuum elements (plane stress, plane strain, axisymmetric, solid). NOCSHE is an input for EAS and it means that no strain considerations will be applied, i.e. no EAS is considered.
To deactivate EAS, click on *Element data 1* in the Geometry browser and in the properties panel click on *Add* and search for NOCSHE.

![Figure 19: Treeview](image)

![Figure 20: Deactivate EAS](image)
We continue by defining the properties to the target and the contactor geometry.
Contact Analysis of a Squeezed Disc | https://dianafea.com

**Main menu** → Geometry → Assign → Shape Properties [Fig. 24]
Shape Properties → Material → Add material [Fig. 25] → Edit material [Fig. 26]
2.3 Boundary Conditions

We define the supports of the model according to Figure 1. However, first we need to define point \( A \). We do this by creating a point and imprint it into the model.

---

**Main menu** ➔ Geometry ➔ Create ➔ Add point ➔ Add point

**Main menu** ➔ Geometry ➔ Modify ➔ Imprint shapes

---

![Add point](image1)

![Imprint point](image2)

![Geometry view](image3)

**Figure 27:** Add point  
**Figure 28:** Imprint point  
**Figure 29:** Geometry view
We continue by adding the supports.

Main menu → Geometry → Assign → Add supports

[Fig. 30] [Fig. 32]
Main menu → Geometry → Assign → Add supports  [Fig. 34]  [Fig. 36]

**Figure 34:** Create support (3)

**Figure 35:** Overview support (3)

**Figure 36:** Create support (4)

**Figure 37:** Overview support (4)
2.3.1 Loads

The loads of this model consist of a prescribed deformation of both plates. The prescribed deformation is the squeezing of the disc by the two plates.

![Figure 38: Application of the top load](image1)

![Figure 39: Application of the bottom load](image2)

![Figure 40: Overview applied loads](image3)
2.3.2 Mesh

We apply mesh seeding to the geometry. Before we remove *Point 1* since we do not need this point anymore. We use 22 divisions for *Circle 1* and *Circle 2* and one division for *Line 1* and *Line 2*. 

![mesh properties - circles](Fig. 41)

![mesh properties - lines](Fig. 42)

Figure 43: Finite element mesh
3 Structural Nonlinear Analysis

3.1 Commands

In this subsection we define the analysis properties as described in Section 1.1. First, we add Analysis1 and a Structural nonlinear command to it. We need to set the geometrically nonlinear analysis since we are working with a contact analysis.

Main menu → Analysis → Add analysis → [Fig. 44] 
Analysis browser → Analysis1 → Add command → Structural nonlinear → [Fig. 47] 
Analysis browser → Analysis1 → Structural nonlinear → Nonlinear effects → Edit properties → [Fig. 46] [Fig. 47]
The first execute block establish the contact process by having a load step of 0.11 m to overcome the gap of 0.1 m. We use a force norm with properties as shown in Figure 52.

![Analysis browser](https://dianafea.com)

**Figure 48: Analysis browser**

**Figure 49: Establish contact process**
Figure 50: Analysis browser

Figure 51: Iteration properties

Figure 52: Force convergence norm
We need to select *Sparse Cholesky* as solution method, since we are dealing with a contact analysis. We add a new execute block and specify the automatic step size settings.

**Analysis browser** ➔ Analysis1 ➔ Structural nonlinear ➔ Solution method ➔ Edit properties

**Figure 53:** Analysis browser

**Figure 54:** Solution method
Analysis browser → Analysis1 → Structural nonlinear → Add... → Execute steps - Load steps [Fig. 55]
Analysis browser → Analysis1 → Structural nonlinear → new execute block 2 → Load steps → Edit properties [Fig. 56] [Fig. 57] [Fig. 58]

Figure 55: Analysis browser
Figure 56: Analysis browser
Figure 57: Load step properties
Figure 58: Automatic step size properties
For this second execute block we work with an energy norm. We use the default value for the convergence tolerance, i.e. 0.001.

Figure 59: Analysis tree

Figure 60: Iteration properties
For the output we select: total translational displacements global, total traction stresses local, total Cauchy stresses global and status contact. For the total traction stresses (STRESS TOTAL TRACTI LOCAL) we select Integration points as location using the properties-button. For the total Cauchy stresses (STRESS TOTAL CAUCHY GLOBAL) we select the option smooth element data values using the properties button. Then, we run the analysis.
3.2 Results

We take a look at the results by displaying displacements TDtXYZ, stresses SZZ and the interface traction in Y direction STy.
Appendix A  Additional Information

Folder: Tutorials/SqueezedDisc

Number of elements \( \approx 200 \)

Keywords:
- ANALYS: geomet nonlin physic.
- CONSTR: contac suppor.
- ELEMEN: contac interf l4ct pstres q8mem t6mem.
- LOAD: deform.
- MATERI: coulom elasti fricti isotro.
- OPTION: adapti direct lagran loadin newton regula size total.
- POST: binary ndiana smooth.
- PRE: dianal.
- RESULT: cauchy contac displa status stress total tracti.
Disclaimer: The aim of this technical tutorial is to illustrate various tools, modelling techniques and analysis workflows in DIANA. DIANA FEA BV does not accept any responsibility regarding the presented cases, used parameters, and presented results.