LARGE DEFORMATION
ZSoil®.PC 070303 report

by
A. Truty
A. Urbański
Th. Zimmermann

Zace Services Ltd, Software engineering
P.O.Box 2, CH-1015 Lausanne
Switzerland
(T) +41 21 802 46 05
(F) +41 21 802 46 06
http://www.zsoil.com,
hotline: hotline@zsoil.com

since 1985
Contents

1 Introduction 3

2 Co-rotational approach 5
   2.1 Co-rotational approach outline ........................................... 5
   2.2 Activation of large deformations option in Z_SOIL code ............... 7
   2.3 Benchmarks for beam elements ............................................. 7
      2.3.1 Euler problem (beam 2D). Eccentric compression in post-buckling range 7
      2.3.2 Curved 3D beam ......................................................... 9
   2.4 Benchmarks for shell elements ............................................. 10
      2.4.1 Cylindrical shell under point forces. ................................ 10
      2.4.2 Spherical shell under point force. Displacement control (post-buckling) 11
   2.5 Benchmarks for continuum elements ..................................... 12
      2.5.1 Euler problem in 2D .................................................. 12
      2.5.2 Euler problem in 3D .................................................. 13

3 Large deformation contact 15
   3.1 Generation of large deformation interfaces ............................. 15
      3.1.1 Setting slave-master attributes ................................... 15
      3.1.2 Generation of large deformation interfaces for evolving structures 17
      3.1.3 Example for generation of contact interfaces in 2D large deformation application - approach I .............................................. 21
      3.1.4 Example for generation of contact interfaces in 2D large deformation application - approach II .............................................. 25
   3.2 Benchmarks for contact interfaces ...................................... 28
      3.2.1 Hertz problem ......................................................... 28
         3.2.1.1 Generation of contact interfaces for Hertz problem .......... 30
      3.2.2 Example of two contacting wheels .................................. 31
Chapter 1

Introduction

ZSoil® 2020 code is designed to handle large rotations and displacements for all structural elements like shells, beams, membranes, anchors and continuum and large deformations as far as contact interface is concerned. The corotational approach is exploited to manage large rotations and its main benefit is that all the stresses and strains at the integrations points remain the engineering ones although are given in the rotated local frame. The new contact formulation, developed to manage really large relative motions of bodies, makes use of so-called slave-master approach in which contacting node (slave) cannot penetrate the corresponding master element face (master) by means of penalty formulation enhanced by Augmented Lagrangian approach (if needed).
Chapter 2

Co-rotational approach

2.1 Co-rotational approach outline

Goal:

to perform large displacement/rotation analysis re-utilizing standard geometrically linear element for:

- beams in 2D and 3D
- shells
- membranes
- truss/anchors

Assumption:

Displacements and rotations attributed to rigid body motion could be arbitrarily large, but "true deformation" remains within small strain limit
CHAPTER 2. CO-ROTATIONAL APPROACH

Performance:

- **deduct rigid body motion** from total deformation of an element, then evaluate element forces and stiffness emerging from "true" deformation.
- Introduce of **element frame E** rigidly attached to the element. Element processing is performed with respect to these frame.
- Deduction of rigid body motion is equivalent to a projection:
  \[ \bar{\delta d} = P \delta d \]
  \[ f = P^T \bar{f} \]
  \[ P = \frac{\partial \bar{d}}{\partial d} \]
  \[ K = P^T K P + \frac{\partial P^T}{\partial d} \bar{f} \]

- Consistent treatment of arbitrarily large rotation:
  * Representation of a rotation by the tensor, use exponential mapping (Rodriguez formula):
    \[ Q = e^\Omega \]
    \[ \Omega = \text{spin}(w) \]
    \[ w = \text{axial}(\Omega) \]
    with:
    \[ Q = e^\Omega = I + \frac{\sin w}{w} \Omega + \frac{1 - \cos^2 w}{w^2} \Omega^2 ; \]
    \[ w = ||w|| \]
  * No additive update, use products of rotation tensors:
    \[ R^{n+1} = \Delta RR^n \]

- Consistent linearization of all force terms.
2.2 Activation of large deformations option in Z_SOIL code

The geometrical nonlinearity option is activated by the check box Large displacement / rotations in the bottom part of the dialog box Analysis and drivers under menu Control / Analysis & Drivers. The check box is active only if the version type is set as Advanced during Control / Project preselection or in the Analysis and drivers dialog.

![Analysis & Drivers dialog box](image)

**Figure 2.2: Analysis & Drivers dialog box**

**Remarks:**
- once this option is activated the whole analysis will be run as geometrically nonlinear
- standard contact elements (segment to segment) cannot be used, only large deformation contact is allowed
- switching ON/OFF this option during restarts will yield computation failure

2.3 Benchmarks for beam elements

2.3.1 Euler problem (beam 2D). Eccentric compression in post-buckling range

**Data file: Euler2D.inp**

The problem of buckling of an elastic beam and tracing its behavior in the post-critical domain (reference: Życzkowski), is analyzed. The geometry, cross section and boundary conditions are shown in figure below. The uniform finite element mesh consisting of 10 equal size beam elements was used in the simulation. Material properties are as follows: $E = 100000$ [kPa], $\nu = 0.0$. This test is run as force driven starting from $N = 0$, $M = 0$ up to 4 times value of the Euler critical force ($N_{crit} = \frac{\pi^2 E I}{(2L)^2} = 2.056$ [kN]) by gradually applying normal force and the moment. The comparison of the reference solution by Życzkowski) and the numerical one is shown in the second figure.
CHAPTER 2. CO-ROTATIONAL APPROACH

Figure 2.3: Euler beam problem. Data, initial and final configurations

Figure 2.4: Euler problem. Force-displacement diagram
2.3.2 Curved 3D beam

Data file: Litewka-Wriggers.inp

The 3D problem of bending of an arch (see figure below) caused by the nodal force \( F = 7\frac{EI}{R^2} = 2916.7 \text{ lbf} \) is analyzed. Geometry, cross section, boundary conditions and the load are shown in the figure below. Material properties are as follows: \( E = 5.10^7 \text{ lbf/in}^2 \), \( \nu = 0.2 \).

![Diagram of 3D Curved Beam](image)

Figure 2.5: 3D Curved beam problem. Geometry, boundary conditions and mesh

The comparison of the reference (Litewka, Wriggers) and numerical results for displacement vector at the point of application of the force is given in the table below.

<table>
<thead>
<tr>
<th></th>
<th>( U_x ) [in] ref. solution</th>
<th>( U_x ) [in] Z_SOIL result</th>
</tr>
</thead>
<tbody>
<tr>
<td>( U_x )</td>
<td>13.5</td>
<td>13.5</td>
</tr>
<tr>
<td>( U_y )</td>
<td>-23.6</td>
<td>-23.6</td>
</tr>
<tr>
<td>( U_z )</td>
<td>54.0</td>
<td>53.3</td>
</tr>
</tbody>
</table>
2.4 Benchmarks for shell elements

2.4.1 Cylindrical shell under point forces.

Data file: Cylinder_free_edgeDC.inp

A cylindrical shell, shown in figure below, loaded by an imposed displacement is analyzed. Mesh, boundary conditions, material and geometrical parameters are also given in the figure. The comparison of force-displacement diagram with the reference solution (Chróścielewski) is given in the next figure.

![Figure 2.6: Free edge cylinder. The data](image)

![Figure 2.7: Free edge cylinder. Displacement history](image)
2.4.2 Spherical shell under point force. Displacement control (post-buckling)

Data file: HingedSphereOnSquare.inp

A spherical shell, shown in figure below, loaded by an imposed displacement applied at the node at the origin of coordinate system is taken into consideration. Geometry of the shell, mesh and boundary conditions are shown in the figure. The comparison of force-displacement diagram, at the node of application of the imposed displacement, versus reference solution after Chróscielewski is given in the next figure.

\[
E = 68950
\]
\[
\nu = 0.3
\]
\[
L / 2 = 0.7849
\]
\[
R = 2.54
\]
\[
h = 0.09945
\]

Figure 2.8: Clamped spherical shell. The data

Figure 2.9: Clamped spherical shell. Load-displacement graph
CHAPTER 2. CO-ROTATIONAL APPROACH

2.5 Benchmarks for continuum elements

2.5.1 Euler problem in 2D

Data file: Euler-2d-continuum.inp

The problem of elastic buckling is analyzed here using exclusively continuum elements in the plane-strain format. The material imperfection is assumed by decreasing the E modulus in a single element shown in the figure. Geometry (L=10m, H=1m), mesh and boundary conditions are shown in figure below. The loading programme is driven by an imposed horizontal displacement applied to the point fixed at the right end of the beam. Material properties are as follows: $E = 100000$ kPa, $\nu = 0.0$. The theoretical buckling force according to the beam theory is equal to $F_{crit} = \frac{\pi^2 E J}{(2 L)^2}$ which yields $F_{crit} = 205.6$ kN/m. The force-deflection diagram is shown in the next figure.

![Material imperfection](image1.png)

**Figure 2.10: Elastic buckling problem in 2D**

![Force-deflection diagram](image2.png)

**Figure 2.11: Force-deflection diagram**
2.5.2 Euler problem in 3D

Data file: Euler-3d-continuum.inp

The problem of elastic buckling is analyzed here using exclusively continuum elements in the 3D format. The material imperfection is assumed by decreasing the E modulus in a single element shown in the figure. Geometry \( L = 10 \text{ [m]}, B = H = 1 \text{ [m]} \), mesh and boundary conditions are shown in figure below. The loading programme is driven by an imposed horizontal displacement applied to the point fixed at the right end of the beam. Material properties are as follows: \( E = 100000 \text{ [kPa]}, \nu = 0.0 \). The theoretical buckling force according to the beam theory is equal to \( F_{\text{crit}} = \frac{\pi^2 E J}{(2L)^2} \) which yields \( F_{\text{crit}} = 205.6 \text{ kN/m} \). The force-deflection diagram is shown in the next figure.

![Elastic buckling problem in 3D](image1)

**Figure 2.12: Elastic buckling problem in 3D**

![Force-deflection diagram](image2)

**Figure 2.13: Force-deflection diagram**
Chapter 3

Large deformation contact

3.1 Generation of large deformation interfaces

3.1.1 Setting slave-master attributes

In the ZSoil® 2020 code we use so-called node-segment elements. The contact interface in large deformations regime can be generated as continuum to continuum, continuum to shell, continuum to beam. So called symmetric contact should be generated always to avoid differences in results once switching between master and slaves is made. However, this is often not possible (beam interfaced with continuum 3D for instance) or not supported so far. All the possibilities in contact generation are given in the following 6 figures.

Figure 3.1: Continuum 2D - Continuum 2D interface setting

Slaves-L ={A,B}
Slaves-R={C,D}
Master-L={ML}
Master-R={MR}
Interface1 = { Slaves-L,Master-R}
Interface2 = { Slaves-R,Master-L}

Nonsymmetric contact: only Interface1 defined or Interface2
Symmetric contact: both Interface1 and Interface2 exist
CHAPTER 3. LARGE DEFORMATION CONTACT

Slaves-L ={A,B}
Slaves-R={C,D}
Master-L={ML}
Master-R={MR}
Interface1 = { Slaves-L,Master-R}
Interface2 = { Slaves-R,Master-L}

Nonsymmetric contact:  only Interface1 defined or Interface2
Symmetric contact: both Interface1 and Interface2 exist

Figure 3.2: Continuum 2D - Beam 2D interface setting (case 1)

Slaves-R={C,D}
Master-L={ML}
Interface2 = { Slaves-R,Master-L}

Nonsymmetric contact: only Interface2 is possible

Figure 3.3: Continuum 2D - Beam 2D interface setting (case 2)

Slaves-L ={A,B,C,D}
Slaves-R={E,F,G,H}
Master-L={ML}
Master-R={MR}
Interface1 = { Slaves-L,Master-R}
Interface2 = { Slaves-R,Master-L}

Nonsymmetric contact: only Interface1 defined or Interface2
Symmetric contact: both Interface1 and Interface2 exist

Figure 3.4: Continuum 3D - Continuum 3D interface setting
3.1. GENERATION OF LARGE DEFORMATION INTERFACES

Continuum 3D vs Beam

Here nonsymmetric contact is the only possibility

Figure 3.5: Continuum 3D - Beam3D interface setting

Continuum 3D vs Shell

Here symmetric contact is possible

Here symmetric contact is not possible

Shell nodes can only be defined as slave nodes and shell itself cannot be a master due to orientation

Figure 3.6: Continuum 3D - Shell interface setting

3.1.2 Generation of large deformation interfaces for evolving structures

The two general approaches concerning generation of contact interfaces (node-segment) for problems in which the geometry is varying in time are implemented in the current version. In the first approach deformations during construction must be small while in the second approach this restrictive assumption is no more valid. In the first approach new contact interface added in the subsequent fill step cancels the initial deformation of the contacting bodies (for instance lining and fill material) and this the reason why it can be used exclusively when initial deformations of the contacting bodies (during construction !) are small enough. In this approach all rules concerning slave-master setting, given in the previous section, are valid.
The second approach is more general but it yields some limitations on slave-master setting. To explain the general idea let us consider an example of excavation followed by lining construction and then filling. The aim of the simulation is to perform an excavation and then construction of a lining followed by seven stages of filling (see Fig. 3.7). After an excavation we get some deformations which we will neglect when a tunnel lining is built. To do that the program memorizes total deformation $\mathbf{U}_e$. This is done automatically (see Fig. 3.8). In the next step the tunnel lining is added and its initial configuration is assumed to be undeformed (just before construction) regardless nonzero deformation at nodes at the bottom slab of the lining. This can be managed thanks to the memorized deformation $\mathbf{U}_e$. The major problem appears when we begin to add fill material. This is so because newly added fill should satisfy contact kinematics if it touches already deformed lining (due to its own dead weight and/or loading imposed during previous fill steps) and existing deformation on the remaining boundaries (see Fig. 3.9). If we consider the situation shown in Fig. 3.10 we can notice that the initial undeformed mesh in the zone of fill (stage I) must be mapped onto deformed configuration caused by a construction of a lining. Hence the boundary nodes along section A-B must satisfy the contact kinematics (cannot penetrate the lining and cannot be separated from it), nodes along the boundary A-D and D-E must fit current deformation equal to $\mathbf{U} - \mathbf{U}_e$ (the one corresponding to the settlement caused by lining construction), and nodes along section E-B must remain at the initial elevation.

![Figure 3.7: Tunnel construction followed by several fill steps](image)

This mapping is made in the code by a finite element solution of a sub-domain subject to the imposed boundary displacements. As the result we get a shift to the nodal coordinates of all newly added continuum elements (NB. in this finite element sub-problem we assume artificial elastic constants $E = 1.0$ and $\nu = 0.0$). However, to make this mapping, all nodes along the section A-B, being part of the contact interface, must be slave nodes (!). It should be emphasized here that incremental deformations during single fill step should be small otherwise strain incompatibilities along the section A-D can cause stress oscillations (although total deformation caused by filling can be large).
3.1. GENERATION OF LARGE DEFORMATION INTERFACES

Figure 3.8: Registering total deformation after excavation

Figure 3.9: Deformation increment due to construction of the lining

Figure 3.10: Mapping of the fill subdomain onto deformed configuration
CHAPTER 3. LARGE DEFORMATION CONTACT

To summarize the following general rules are used in the second approach:

- structures (beams/truss/membranes/shells) are always added in undeformed configuration;
  hence, whenever new structural element is added its initial total deformation must be
  memorized as $U_{os}$ (at the element level); the current structure deformation is always equal
  to $U - U_{os}$
- continuum is added after mapping to the deformed configuration
- at the end of the excavation time step the current total deformation corresponding to that
  state is memorized as $U_e$ for all nodal points

Mapping from undeformed to the deformed configuration is performed by imposing displace-
ments at the boundary nodes according to the following rules:

- if node is of SLAVE type then we project it on master in the following manner
  if master is a structural element then impose on slave node a shift equal to $U - U_{os}$
  if master is not a structural element impose on slave node a shift equal to $U - U_e$

- if node is not of a SLAVE type then
  if node belongs to the structural element existing at time $t_N$ then set shift $U - U_{os}$
  if node does not belong to the structural element existing at time $t_N$ then set shift $U - U_e$

- if node has a solid BC on certain DOF then impose them
- if node is on the free external boundary impose zero deformation in y direction

NB. Activation of the approach II has be done in the dialog Control /Analysis and drivers
by switching the Large displacement /rotations ON. Then under Settings, in the appearing
dialog checkbox Update coordinates during costruction has to be set ON, otherwise algorithm
I will be performed

Figure 3.11: Activating algorithm II
3.1.3 Example for generation of contact interfaces in 2D large deformation application - approach I

Data file: test-frame-filling.inp

Let us consider an example of an excavation and then tunnel construction followed by filling. The main problem is on how to generate contact interface starting with mesh as shown in Fig. (3.12). To do that we have to disconnect the mesh along the interface contour. In this example all nodes which belong to the continuum elements adjacent to the lining and being external to the lining have new node numbers. This operation can be performed in the following manner:

1. select all quads inside lining
2. select all beams (see Fig. (3.13)
3. remove selected elements virtually (hide them)
4. select nodes to be duplicated (see Fig. (3.14)
5. under menu Interfaca(large deformation) use method Create new nodes on selected.
   at that moment we have duplicated nodes along the lining and new nodes belong to the continuum elements adjacent to beams (see Fig. (3.15)

Once we have disconnected the mesh along the contour we can generate contact interface in the following way

1. highlight contact element contour (edges which will play a role of so-called masters) (see Fig. (3.16), and create masters with a label ”masters”
2. select part of the lining (beam elements) (see Fig. (3.17) and then select nodes from selected beam elements
3. create group of slave nodes (contactors) with a label ”slave nodes on beam”
4. using an option Create\Update contact elements define the interface by merging the pair ”slave nodes on beam”-”masters” (see Fig. (3.18, Fig. (3.19)

Figure 3.12: Mesh and distribution of existence functions

EXF=2
EXF=3
EXF=4
EXF=5
EXF=1
EXF=6  for beams
EXF=7  for interface
CHAPTER 3. LARGE DEFORMATION CONTACT

Figure 3.13: Selection of the lining interior

Figure 3.14: Selection of nodes to be duplicated
3.1. GENERATION OF LARGE DEFORMATION INTERFACES

Figure 3.15: Nodes after duplication procedure

Figure 3.16: Generation of masters
CHAPTER 3. LARGE DEFORMATION CONTACT

Figure 3.17: Selection of nodes to set them as slaves

Figure 3.18: Generation of slave nodes
3.1. GENERATION OF LARGE DEFORMATION INTERFACES

3.1.4 Example for generation of contact interfaces in 2D large deformation application - approach II

Data file: test-frame-filling-n.inp

The same example analyzed with the second approach requires a different setting for the interface. The details of the data generation are given in the following 5 figures.
CHAPTER 3. LARGE DEFORMATION CONTACT

Masters set on continuum elements inside the lining „masters-continuum”

Figure 3.21: Location of masters to handle continuity condition before excavation

Masters set on beam elements: „new-masters”

Figure 3.22: Location of masters to handle filling
3.1. GENERATION OF LARGE DEFORMATION INTERFACES

Figure 3.23: Setting contact interface parameters for interface structure-fill

Figure 3.24: Setting continuity interface along initial position of the lining contour
3.2 Benchmarks for contact interfaces

Data preparation in this case is somewhat different compared to the standard contact interface setting known for small deformation applications. In the large deformations regime slave nodes may interact with different master faces during the analysis hence explicit setting of so-called contact elements is not possible. In general we may associate the attribute "contactor node" with any node in the mesh and another attribute which is the "master face" with any face of the finite element (continuum, shell, beam (only 2D) or membrane).

3.2.1 Hertz problem

Data files: hertz-2d-dense-foundation, hertz-2d


\[
F = 500 \text{ kPa}, \quad \nu = 0.3 \\
q = \frac{F}{2R} \\
R = 8m \\
E = 500 \text{ kPa}, \quad \nu = 0.3 \\
E = \infty
\]

![Figure 3.25: Hertz problem setting](image)

As the foundation is to be rigid we can generate the mesh as shown in figure 3.26. However, contact stress recovery is not that easy matter in the node-segment contact implementations. Hence we can generate different mesh for the foundation with which, using standard tools available in the postprocessor (cross sections through the mesh), we can easily recover the interesting values. This second mesh is shown in figure 3.27.

The comparison of normal stresses in the interface for two levels of the force \( F = 50 \text{ kN/m} \) and \( F = 100 \text{ kN/m} \) \( q = \frac{F}{2R} \) is presented in figure 3.28.
3.2. BENCHMARKS FOR CONTACT INTERFACES

Figure 3.26: Hertz problem in plane strain format

Figure 3.27: Hertz problem in plane strain format - dense mesh in the foundation

Figure 3.28: Hertz problem: Distribution of contact stresses
3.2.1.1 Generation of contact interfaces for Hertz problem

The following figures explain on how to create contact interfaces in the 2D applications.

Figure 3.29: In menu INTERFACE (LARGE DEFORMATIONS) create group of slave nodes (contactors)

Figure 3.30: In menu INTERFACE (LARGE DEFORMATIONS) create group of master faces
3.2.2 Example of two contacting wheels

Data file: two-wheels.inp

The aim of this example is to show some details concerning generation of the large deformation contact interfaces. The important thing is that in cases of the two deformable bodies we should generate so-called symmetric contact in which boundaries of the two bodies have both slave nodes and master faces. This test is driven by applied vertical displacement.
CHAPTER 3. LARGE DEFORMATION CONTACT

Figure 3.32: Finite element meshes for two wheels and foundation

Figure 3.33: Setting slave nodes (contactors) on left wheel
3.2. BENCHMARKS FOR CONTACT INTERFACES

Figure 3.34: Setting slave nodes (contactors) on right wheel

Figure 3.35: Setting master faces on foundation mesh
CHAPTER 3. LARGE DEFORMATION CONTACT

Figure 3.36: Setting master faces on left wheel

Figure 3.37: Setting master faces on right wheel
3.2. BENCHMARKS FOR CONTACT INTERFACES

Figure 3.38: Generation of interface left wheel-foundation

Figure 3.39: Generation of interface left wheel-right wheel
CHAPTER 3. LARGE DEFORMATION CONTACT

Figure 3.40: Generation of interface right wheel-foundation

Figure 3.41: Generation of interface right wheel-left wheel
3.2. BENCHMARKS FOR CONTACT INTERFACES

Figure 3.42: Deformation at stage of first contact of the two wheels

Figure 3.43: Deformation at stage of advanced contact of the two wheels
CHAPTER 3. LARGE DEFORMATION CONTACT

Figure 3.44: Deformation at stage of separation of the wheels from the foundation (at central point)

Figure 3.45: Deformation at stage of advanced separation of the wheels from the foundation
Figure 3.46: Evolution of vertical displacement
REFERENCES


