APPENDIX 1

INDENTATION OF RIGID SPHERE ON A ELASTIC-PLASTIC PLATE

A1.1 INTRODUCTION

Indentation of a elastic-plastic half-space by a rigid sphere is of great importance in various material processes and mechanics applications. The macroscopic simulation of indentation of a rigid sphere and a half-space, leads to the full visualization of the distribution of contact stresses between those bodies. The important characteristics of the problem are the localized deformation and the variation in the contact area with contact force and the resultant high stresses due to the nonlinear relationship between the contact force and displacement, which leads to plastic deformation in most engineering contact problems.

A finite element analysis of frictionless indentation of a half-space by a rigid sphere is presented and the deformation behavior during loading is examined in terms of the interference and elastic–plastic material properties. The analysis yields dimensionless constitutive relationships for the normal load, contact area, and mean contact pressure during loading for a wide range of material properties and interference ranging from the inception of yielding to fully plastic deformation.
A1.2 THEORY OF ELASTIC-PLASTIC CONTACT

The elastic–plastic indentation of a homogeneous half-space by a rigid sphere is a fundamental problem in contact mechanics and of great importance in various material processes and mechanics applications, such as powder compaction, contact of rough surfaces, and thermal and electrical conductance. In powder compaction, the prediction of the global force-displacement behavior requires knowledge of the local indentation response between the particles. Likewise, analyses of contact deformation, surface adhesion, and static and dynamic friction of rough surfaces are based on single-asperity constitutive relationships of the contact parameters.

Indentation has been used since the beginning of the previous century to evaluate the plastic properties of metals. The elasticity of real materials plays an important role in the plastic indentation process. When the yield is first exceeded the plastic zone is small and fully contained by material, which remains elastic so that the plastic strains are of the same magnitude as the surrounding elastic strains. The material displaced by the intender is accommodated by the elastic expansion of the surrounding solid. As the indentation becomes more severe, either by increasing the load on a curved intender, an increasing pressure is required beneath the indenter to produce necessary expansion. Eventually the plastic zone breaks out to the free surface and the displaced material is free to escape by plastic flow to the sides of the intender. This is the uncontained mode of deformation analyzed by theory of rigid plastic solids. By which the uncontained mode is possible when the pressure beneath the intender reaches the value \( p_m = cY \) where \( c \) has a value about 3.0 depending upon the geometry of the intender and the friction at the interface. First yield at which \( c \) has a value about unity. There is a transitional range of contact pressures, lying between \( Y \) and \( 3Y \), where the
elastic material contains the plastic flow and the mode of deformation is one of roughly radial expansion.

The three ranges of loading: Pure elastic, elastic-plastic (contained) and fully plastic are a common feature of most engineering structures. The stress strain relations of Reuss govern the deformation of elastic perfectly plastic material. In principle the contact stresses due to an elastic plastic indentation in which the strains remain small can be calculated. In practice it is very difficult because the shape and size of the elastic-plastic boundary is not known a priori. Numerous experiments on the normal indentation of an elastic plastic half space by a rigid sphere were conducted in the early 1950’s. The results showed that, once the contact force exceeds a certain value, the distribution of contact pressure is almost uniform over most of the contact area. This was confirmed by FEA results. Hardy et al (1971) showed that the Hertz pressure distribution is valid until \( p_0 = 1.6Y \), when the yield stress \( Y \) was reached below the center of the contact area.

Further indentation resulted in spreading of the plastic deformation zone below the surface and a slight modification of the shape of the contact pressure distribution as the maximum pressure increased further. When the contact force was about 6 times the contact force at initial yield, the plastic deformation zone in the substrate reached the contact surface at the perimeter of the contact area. Beyond this point, further indentation resulted in dramatic change in the pressure distribution: over an enlarging central portion of the contact area, the contact pressure was almost constant with only slight increase in the maximum contact pressure. Once the pressure reaches 2.8-3.0 times the yield stress, the pressure distribution is almost flattened and further increase in contact force results only in enlarging the contact area while the maximum pressure does not appear to increase further.
A1.3 CONTACT ANALYSIS USING ANSYS

The ANSYS finite element analysis (FEA) program offers a variety of elements designed to treat cases of changing mechanical contact between the parts of an assembly or between different faces of a single part. These elements range from simple, limited idealizations to complex and sophisticated, general-purpose algorithms. FEA analysts are frequently faced with modeling situations where changing contact cannot be assumed negligible and ignored. Finding the best choices for contact elements, element options, solver, and solution options can drastically improve the model’s performance and reduce the analyst’s frustration with a contact simulation model.

A1.3.1 Difficulties in the Contact Problem

Despite the importance of contact in the mechanics of solids and its engineering applications, contact effects are rarely taken into serious account in conventional engineering analysis, because of the extreme complexity involved. Mechanical problems involving contacts are inherently nonlinear. Bodies in contact may have complicated geometries and material properties and may deform in a seemingly arbitrary way. With the rapid development of computational mechanics, however, great progress has been made in numerical analysis of the problem. Using the finite element method, many contact problems, ranging from relatively simple ones to quite complicated ones, can be solved with high accuracy. The Finite Element Method can be considered the favorite method to treat contact problems, because of its proven success in treating a wide range of engineering problem in areas of solid mechanics, fluid flow, heat transfer, and for electromagnetic field and coupled field problems.
A1.3.2 Contact Problem Classification

There are many types of contact problems that may be encountered, including contact stress, dynamic impacts, metal forming, bolted joints, crash dynamics, assemblies of components with interference fits, etc. All of these contact problems, as well as other types of contact analysis, can be split into two general classes in ANSYS

1. Rigid - to - flexible bodies in contact,
2. Flexible - to - flexible bodies in contact.

In rigid - to - flexible contact problems, one or more of the contacting surfaces are treated as being rigid material, which has a much higher stiffness relative to the deformable body it contacts. Many metal forming problems fall into this category. Flexible-to-flexible is where both contacting bodies are deformable. Examples of a Flexible-to-flexible analysis include gears in mesh, bolted joints, and interference fits.

A1.3.3 Types of Contact Models

In general, there are three basic types of contact modeling application as far as ANSYS use is concerned.

Point-to-point contact: the exact location of contact should be known beforehand. These types of contact problems allow only small amounts of relative sliding deformation between contact surfaces.

Point-to-surface contact: the exact location of the contacting area may not be known beforehand. These types of contact problems allow large amounts of deformation and relative sliding. Also, opposing meshes do not need to have the same discretisation or a compatible mesh.
**Surface-to-surface contact:** It is typically used to model surface-to-surface contact applications of the rigid-to-flexible classification.

### A1.3.4 Basic Steps in a Contact Analysis

The basic steps for performing a typical surface-to-surface contact analysis are listed below.

1. Create the model geometry and mesh
2. Identify the contact pairs
3. Designate contact and target surfaces
4. Define the target surface
5. Define the contact surface
6. Set the element KEYOPTS and real constants
7. Define/control the motion of the target surface
8. Apply necessary boundary conditions
9. Define solution options and load steps
10. Solve the contact problem
11. Review the results

### A1.3.5 Simulating contact between parts

FEA program in ANSYS offers a variety of elements designed to treat cases of changing mechanical contact between the parts of an assembly or between different faces of a single part. These elements range from simple, limited idealizations to complex and sophisticated, general, purpose algorithms. FEA analyses are frequently faced with modeling situations where changing contact cannot be assumed negligible and ignored. Finding the best choices for contact elements, element options, solver and solution options can drastically improve the models performance and reduce the analyst’s frustration with a contact simulation model.
A1.3.6 Method of solving contact problems

There are two methods of satisfying contact compatibility:

1. A penalty method, and
2. A combined penalty plus a Lagrange multiplier method.

The penalty method enforces approximate compatibility by means of contact stiffness. The combined penalty plus Lagrange multiplier approach satisfies compatibility to a user-defined precision by the generation of additional contact forces that are referred to as Lagrange forces. It is essential to prevent the two areas from passing through each other. This method of enforcing contact compatibility is called the penalty method. The penalty allows surface penetrations, which can be controlled by changing the penalty parameter of the combined normal contact stiffness. If the combined normal contact stiffness is too small, the surface penetration may be too large, which may cause unacceptable errors. Thus the stiffness must be big enough to keep the surface penetrations below a certain level. On the other hand, if the penalty parameter is too large, then the combined normal contact stiffness may produce severe numerical problems in the solution process or simply make a solution impossible to achieve.

A1.3.7 Contact element types

Contact elements are constrained against penetrating the target surface. However, target elements can penetrate through the contact surface. For rigid-to-flexible contact, the designation is obvious: the target surface is always the rigid surface and the contact surface is always the deformable surface. Contact elements can be grouped into four general categories based on increasing levels of sophistication or complexity:
• Point-to-point gap elements:
  CONTACT12, LINK10, COMBIN40, CONTAC52, CONTA178

• Point-to-line contact elements:
  CONTAC26

• Point-to-surface contact elements:
  CONTAC48, CONTAC49

• Surface-to-surface contact elements:
  TARGE169, TARGE170, CONTA171, CONTA172, CONTA173, CONTA174

A1.3.8 Contact Element Advantages, Disadvantages and their Convergence

Because of the simplicity of their formulation, the advantages of using Contact elements are:

1. They are easy to use
2. They are simple to formulate, and
3. They are easily accommodated into existing FE code.

However, contact elements pose some difficulties such as the fact that their performance, in term of convergence and accuracy, depends on user-defined parameters. In order to get convergence in ANSYS, difficult problems might require many load increments, and if much iteration is required, then the overall solution time increases. Balancing expense versus accuracy: All FEA involves a trade-off between expense and accuracy. More detail and a finer mesh generally leads to a more accurate solution, but require more time and system resources. Nonlinear analyses add an extra factor, the number of load increments, which affect both accuracy and expense. Other nonlinear
parameters, such as contact stiffness, can also affect both accuracy and expense. One must use own engineering judgment to determine how much accuracy is needed versus how much expense can be afforded.

Using contact elements in an FEA simulation is seldom a simple, painless experience. Mathematical theories only describe the stresses in the contact region. Often stresses away from the contact zone are to be studied, and the contact behaviour must be modeled properly to find the contact stress. Proper computation of the forces and deformations in the contact zone is critical to determining the stress results through the model. As an example in gear teeth contact maximum stress expected at the root of gear tooth and not under the contact region.

### A1.4 ELASTIC PLASTIC INDENTATION

Figure A1.1 shows a rigid sphere of radius R indenting a half-space. The interference ‘\( \omega \)’ and contact radius ‘\( a \)’ correspond to a normal load P. The displacement of the contact edge measured from the original surface h is assumed to be positive if the material deforms as shown in the Figure A1.1. The mean contact pressure \( p_m \) is defined as,

\[
p_m = \frac{P}{A}
\]  

(A1.1)

where A is the contact area, For dimensionless analysis, the interference, contact area, and mean contact pressure are normalized by the contact radius a, and material yield strength Y, respectively.
For small interference, the indentation response is elastic, and according to Hertz Theory

$$p_m/Y = \frac{3 (E^a/Y)}{\pi R}$$  \hspace{1cm} (A1.2)

where $E^*$ is the equivalent elastic modulus, given by equation (4.5)

$E_1$ is replaced by $E^*$ to combine the two elastic properties (i.e., $E_1$ and $\nu_1$) into one elastic parameter, as suggested by Hertz (1882). Experiments and numerical results have shown that the equivalent elastic modulus adequately describes the elastic contribution to the overall deformation in the elastic–plastic deformation regime. It has been shown that when the dimensionless interference reaches a critical value, yielding takes place at which the mean contact pressure is $p_m/Y = 1.07$. Further increase of the interference produces an elastic–plastic indentation response affected by plastic flow. The inception of fully plastic deformation is encountered at the instant that the mean contact pressure reaches a maximum for the first time. This peak value of the mean contact pressure is related to the material

![Figure A1.1 Rigid Sphere Indenting A Half-Space](Ref: Kogut and Komvopoulos 2004)
hardness Johnson (1985) claimed that deformation in elastic–plastic indentation depends on the ratio of the representative strain below the indenter a/R to the yield strain of the half-space Y/E. Similar to the elastic Hertz solution, two elastic properties are combined into one elastic parameter, and the indentation is characterized by a single dimensionless parameter E^*a/YR.

**A1.5 CONTACT BETWEEN A RIGID SPHERE AND A HALF-SPACE**

In the contact interaction between a rigid sphere and a half space, if the maximum stress generated in the substrate by the contact force remains below the yield stress, the contact is purely elastic and the problem can be described by the hertz theory in which the pressure is elliptically distributed over the contact surface and the displacement of the center of the sphere is proportional to the square of the contact radius a and inversely proportional to the radius of the sphere R.

\[
p(r) = p_0 \{1-(r/a)^2\}^{1/2} \tag{A1.3}
\]

\[
p_0 = 2E^* a/\pi R = 2Ea/\pi (1-v^2)R \tag{A1.4}
\]

\[
\omega = a^2 / R \tag{A1.5}
\]

where \(p(r)\) is the contact pressure distribution over the contact surface. The contact force \(P\) is obtained by integration of the pressure over the contact area,

\[
P = (4/3)E^* R^{1/2} \omega^{3/2} \tag{A1.6}
\]

**A1.6 FINITE ELEMENT MODELING**

The finite element simulations were performed with the multi-purpose code ANSYS. The following assumptions were used in the finite element analysis:
1. Perfectly smooth surfaces,
2. Frictionless contact,
3. Homogeneous, isotropic half-space
4. Adhesion less contact interface.

Using ANSYS an Axisymmetric 2-D model is created. The rigid contact surface option was used to simulate the rigid indenter, which is modeled by a curved line. The half-space was modeled with axisymmetric, eight-node, quadratic, Isoparametric elements. Contact between the indenter and the half-space surface was detected by surface-to-surface contact elements. The mesh consists of approximately 10128 elements. The model is meshed densely in contact region to detect contact area accurately. The contact region is meshed using higher order surface-to-surface contact elements. The nodes on boundaries $y = 0$ and $x = 0$ were constraint against displacement in the $x$ and $y$ direction, respectively. The resulting meshed model is shown in the Figure A1.2.

![Rigid sphere](image)

**Figure A1.2** Representative model for indentation analysis (Ref: Kogut and Komvopoulos 2004)
A1.7 FINITE ELEMENT ANALYSIS

Due to the nonlinear material behavior (plasticity), geometric nonlinearity (i.e., large displacements), and surface contact, an updated Lagrangian formulation is used in the finite element analysis. For calculating the critical interference CEB model is used which approximates elastic-plastic contact by modeling a plastically deformed portion of a hemisphere using volume conservation. The critical interference that marks the transition from elastic to elastic-plastic deformation regime given by CEB model is given in equation (4.7) as

\[ \eta = \left( \frac{\pi KH}{2E} \right)^2 R \]

There are two ways to simulate the contact problem. The first applies force to the body and then computes the resulting displacement. The second applies the displacement and then computes the resulting contact force. In both methods displacement, stress and the strain in the body can be determined. In this analysis the latter approach is used, where the nodes of sphere is displaced of particular interference and the resulting contact parameters are calculated. This method is used because the resulting solution converges more rapidly than the former.

To obtain generalized solutions and to eliminate the dependence of the results on input parameters, the global contact parameters and material properties were used in dimensionless form. The validity of this normalization was evaluated by comparing finite element solutions obtained for different values of \( R, E, \) and \( Y \). The numerical results are in good agreement with the theoretical solution in the elastic regime and the maximum difference is less than 1%. Yield starts at a point located below the center of the contact area, as
predicted by the Hertz theory, which indicates that the finite element model and correctness of the assumed boundary conditions.

A1.7 FINITE ELEMENT MODEL VALIDATION

To validate the finite element discretization and analysis procedure before performing the finite element analysis for elastic-plastic problem, the finite element model is solved for elastic contact problems and results are compared with Hertz elastic model for contact of a rigid sphere and a half-space. To solve the elastic contact problem the finite element model is applied with elastic material property of steel with young’s modulus $E = 2 \times 10^5$ N/mm$^2$ and Poisson ratio 0.3. The finite element model is solved without considering yield criterion and plastic deformation. In all the numerical simulations that were performed with the perfect slip condition at the contact, it was found that during the loading stage the radial displacements of the contacting points are negligibly small compared to their corresponding axial displacements. This suggests that in general there may be a very little tendency to slip at the contact interface, and hence, the assumption of perfect slip condition at the contact may not restrict the generality of the solution.

![Figure A1.3 Interference vs Contact radius](image)

Figure A1.3 Interference vs Contact radius
The Figures A1.3, A1.4 and A1.5 shows the comparison of FEA results with Hertz results. Contact load, contact pressure and contact area are calculated for interference less than critical interference and compared with Hertz results.
A1.8 Finite Element Analysis Results

From the finite element analysis results the following plots are obtained in ANSYS for the interference ratio \( \frac{\omega}{\omega_c} \) values 1 (inception of yield) and 70 (Fully plastic).

Figure A1.6 Axisymmetric 2D model with applied Boundary Conditions

Figure A1.7 Von Mises stress of model at interference \( \frac{\omega}{\omega_c} = 1 \)
A1.9 RESULTS AND DISCUSSION

The results of the FEA are presented for a wide range of interference values. The material properties used for analysis, cover a wide range of applications of steel and aluminum materials of engineering use. The
properties of the material that are taken into consideration are listed in Table A1.1.

![Graph showing the relationship between contact area and interference](image)

**Figure A1.10 Contact Area A/A_c versus Interference \( \frac{\omega}{\omega_c} \)**

By curve fitting the numerical results from finite element analysis the following constitutive relationship between the contact area and interference was obtained for the elastic–plastic deformation regime. The Figure A1.10 shows the relationship between parameters graphically.

\[
A/A_c = 1.03 \left( \frac{\omega}{\omega_c} \right)^{1.1234}
\]  
(A1.7)
Figure A1.11 Contact Load \( P/P_c \) versus Interference \( \frac{\omega}{\omega_c} \)

By curve fitting the numerical results from finite element analysis the following constitutive relationship between the contact load and interference was obtained for the elastic–plastic deformation regime. The Figure A1.11 shows the relationship between parameters graphically.

\[
P/P_c = 1.66 \left( \frac{\omega}{\omega_c} \right)^{1.2418} - 0.61 \quad (A1.8)
\]

According to Johnson first yield for a spherical indentation occurs at \( p_m = 1.1 \, Y \), but from FEA yielding commences when \( p_m/Y = 1.07 \), as predicted by the Hertz theory. This is the lower bound of the elastic–plastic deformation regime. The upper bound of the elastic–plastic deformation regime is obtained at the inception of fully plastic deformation, determined by
the interference at which the mean contact pressure reaches its maximum value 2.8Y.

![Graph showing the relationship between dimensionless interference and mean contact pressure.](image)

**Figure A1.12 Mean contact pressure $p_m/Y$ versus Interference $\frac{\omega}{\omega_c}$**

By curve fitting the numerical results from finite element analysis the following constitutive relationship between the mean contact pressure and interference was obtained for the elastic–plastic deformation regime. The Figure A1.12 shows the relationship between parameters graphically.

$$p_m/Y = \frac{3.507\left(\frac{\omega}{\omega_c}\right)^2 + 18.16\left(\frac{\omega}{\omega_c}\right) - 2.668}{\left(\frac{\omega}{\omega_c}\right)^2 + 13.09 \left(\frac{\omega}{\omega_c}\right) + 3.511}$$

(A1.9)

The results obtained from finite element analysis are more accurate because of the finer mesh of model and larger number of numerical data obtained in the interference range where the mean contact pressure reaches a
maximum 2.8Y. The results obtained at the boundary of the elastic–plastic and fully plastic deformation regimes in terms of a dimensionless parameter yields \( 2.81 \leq \frac{E^*a}{YR} \leq 28.5 \) for \( 200 \leq \frac{E}{Y} \leq 1000 \). This shows that the dimensionless parameter \( (E^*a/YR) \) given by Johnson (1985) is adequate to determine the evolution of deformation in the elastic–plastic deformation regime and the elastic–plastic response over a wide range of material properties. The value used by Johnson to determine the inception of fully plastic deformation \( E^*a/YR \approx 40 \), but from the finite element results shown in the Table 6.1 gives it as,

\[
E^*a/YR = 28.5
\]  

(A1.10)

<table>
<thead>
<tr>
<th>Case</th>
<th>Young’s modulus, (E) in Gpa</th>
<th>Yield stress (Y) in Mpa</th>
<th>Sphere Radius (R) in mm</th>
<th>E/Y Ratio</th>
<th>E*a/YR</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>200</td>
<td>210</td>
<td>5</td>
<td>952.4</td>
<td>27.799</td>
</tr>
<tr>
<td>2</td>
<td>70</td>
<td>100</td>
<td>5</td>
<td>700.0</td>
<td>28.067</td>
</tr>
<tr>
<td>3</td>
<td>80</td>
<td>220.0</td>
<td>1</td>
<td>363.6</td>
<td>28.426</td>
</tr>
<tr>
<td>4</td>
<td>200</td>
<td>560.8</td>
<td>8</td>
<td>356.6</td>
<td>28.488</td>
</tr>
<tr>
<td>5</td>
<td>106</td>
<td>357</td>
<td>10</td>
<td>296</td>
<td>28.529</td>
</tr>
<tr>
<td>6</td>
<td>200</td>
<td>911</td>
<td>5</td>
<td>219</td>
<td>28.007</td>
</tr>
</tbody>
</table>

A1.10 CONCLUSION

Finite element simulation of a half-space indented by a rigid sphere based on constitutive laws established for the relevant elastic–plastic deformation regime was carried out to examine the effect of the contact parameters on the indentation response during loading. The validity of the axisymmetric finite element model was verified through favorable
comparisons with the Hertz solution. Simple analytical expressions that extend the classical Hertz solution up to the fully plastic deformation regime were derived for a range of material properties. General solutions that are independent of specific material properties and radius of the spherical indenter were obtained based on a normalization scheme.

The dimensionless parameter $E^*a/YR$ used to determine evaluation of elastic plastic deformation regime of a half-space intended by a rigid sphere is found to be 28.5. Finite element analysis results were compared with that of Johnson (1985) results.