Influence of the main contact parameters in finite element analysis of elastic bodies in contact

Gabriel Hattori[1,a*] and Alberto Luiz Serpa[2,b]

[1]School of Engineering & Computing Sciences, Durham University, DH1 3LE, Durham, UK

[2]Department of Computational Mechanics, Faculty of Mechanical Engineering, University of Campinas - UNICAMP, 13083-970 Campinas, SP, Brazil

[a]gabriel.hattori@durham.ac.uk, [b]serpa@fem.unicamp.br

Keywords: Contact parameters, Finite element, Connecting-rod, ANSYS.

Abstract. One of the key issues in solving contact problems is the correct choice of the contact parameters. The contact stiffness, the penetration limit and the contact algorithm are some of the parameters that have to be adjusted. There are no methodologies available in the literature for choosing the contact parameters, relying only on the user experience for this important task. In this work we investigate how the contact parameters behave in a commercial finite element analysis software. We will show that while the contact stiffness has great influence on the finite element analysis, other parameters will not affect it significantly. Some contact examples are shown to illustrate the performance of the contact parameters during the solution of a contact problem.

Introduction

Elastic contact is a non-linear problem which presents several particularities. The main difficulty of this class of problems is to obtain a model that expresses correctly the stress and displacements distributions, both necessary in order to determinate the boundary conditions of the problem. Moreover, the contact region is not previously known, and it may change due to load changes during the analysis.

Most of the works in the contact field consider new algorithms or specific techniques to solve this kind of problem. Some works have studied the influence of contact parameters in the solution of a contact problem, usually with combinations between finite element method and mathematical programming methods, such as the Augmented Lagrangian Method, where the Lagrange multipliers have physical meaning of the contact pressures [1], and the Penalty Method [2], where the penalisation parameters act as the contact stiffness.

In [3], the mortar method is introduced, which consists in a contact region discretisation method. It is shown its improvement compared to the other contact formulations. More recently, a higher-order contact element based on the mortar method was proposed by [4].

A formulation of the contact problem from variational methods is presented using energy methods by [5, 6, 7]. The approach is done through potential energy minimisation with constrained optimisation methods.

Another approach to the contact problem is shown by [8], where minimal principles without an explicit connection with variational methods are applied directly in the contact problem. The concept of conditionally dependent constraints is used to deal with sticking or sliding situations.

A comparison of different implementations of the Augmented Lagrangian Method to the contact problem is presented in [9]. An analysis in terms of variational inequalities of the contact problem is done, and a dual algorithm that permits the update of the penalisation coefficient is proposed.

In [10], a semi-smooth Newton method is combined with the Augmented Lagrangian and the path-following method to solve the contact problem in linear elasticity is discussed. A regularisation technique is applied, allowing an infinite-dimensional analysis, leading to an algorithm with a very fast local converge rate.

This work was motivated by the automotive connecting rod small end contact problem. This component is highly critic: it connects the piston to the crankshaft, transforming the engine energy into torque. Therefore, the durability of this component is extremely important. There are few works that treat this mechanical contact problem, like [11], where it is shown that the contact region and sticking/sliding conditions for some applied loads are approximately the same than the receding contact solution, i.e., there is no clearance nor interference between the contact bodies. In [12], some papers concerning connecting rod small end have analysis with different results, showing that is required more investigation in this topic. The idea of this paper is to develop a methodology to study the connecting rod small end contact problem using the commercial software ANSYS.

Recently, the authors [13] have proposed a method for estimating the contact stiffness factor using artificial neural networks. Some of the contact parameters within ANSYS software were analysed in this previous work. However, in this work we present a more detailed analysis of the influence of several contact parameters, namely the normal contact stiffness, the penetration limit, the contact algorithms and the finite element solvers. Each one of these parameters has a default configuration, attributed at the definition of the contact pair, that may be adequate or not. It is necessary to identify the influence of each factor in contact analysis, contributing to reduce the use of the intuition and trial-and-error approaches to solve the contact problem. Since the analysis of the contact parameters can be extensive, we are not taking into account the influence of friction in the contact problem in this work.

The remaining of this paper is organised as follows: in Section 2 a brief description of the contact problem is presented. Next, Sections 3, 4, 5 and 6 describe the finite element solvers, the normal contact stiffness, the penetration limit and the contact algorithms, respectively. Section 7 shows some numerical results for 2D and 3D contact problems. The conclusions are presented in Section 8.

Contact parameters

The static contact problem can be formulated as a constrained minimisation problem, where the objective function is the potential energy of all bodies in contact, and the constraints are given by the non-penetration condition. Hence, the contact problem can be stated as:

$$\begin{aligned} min \quad & \Pi(\mathbf{u}) = \frac{1}{2}\mathbf{u}^t\mathbf{K}\mathbf{u} - \mathbf{f}^t\mathbf{u} \\ subject\ to \quad & g_j(\mathbf{u}) \leq 0, \quad j = 1, \cdots, n \end{aligned} \tag{1}$$

where $\mathbf{u}$ is the displacement vector (optimisation variable) and $g_j(\mathbf{u})$ is the non-penetration constraint, with $g_j(\mathbf{u}) = 0$ defining the bodies in contact.

In a problem with two elastic bodies, A and B, the total potential energy is defined as:

$$\Pi(\mathbf{u}) = \Pi_A(\mathbf{u}) + \Pi_B(\mathbf{u}) = \frac{1}{2}\left\{\begin{array}{c}\mathbf{u}_A\\ \mathbf{u}_B\end{array}\right\}^t \left[\begin{array}{cc}\mathbf{K}_A & \mathbf{0}\\ \mathbf{0} & \mathbf{K}_B\end{array}\right] \left\{\begin{array}{c}\mathbf{u}_A\\ \mathbf{u}_B\end{array}\right\} - \left\{\begin{array}{c}\mathbf{f}_A\\ \mathbf{f}_B\end{array}\right\}^t \left\{\begin{array}{c}\mathbf{u}_A\\ \mathbf{u}_B\end{array}\right\} \tag{2}$$

Fig. 1 illustrates the physical contact problem, where there is no penetration between the elastic bodies, and contact reaction forces arises in the contact area. Fig. 2 represents the numerical contact problem. In this case, the usual approach is to consider that the bodies are in contact when non-penetration constraints are not satisfied. The non-penetration constraints are then redefined as $g(u) \leq TOLN$, where $TOLN$ is the acceptable penetration level for the contact analysis.

Some algorithms of constrained minimisation can be used to solve the contact problem. The most common are: Penalty Method, Lagrange Multipliers Method and Augmented Lagrangian Method. Some parameters in these methods may have a physical interpretation (like the Lagrange Multipliers,

$$
\begin{aligned}
min \quad & \Pi(\mathbf{u}) = \frac{1}{2}\mathbf{u}^t\mathbf{K}\mathbf{u} - \mathbf{f}^t\mathbf{u} \\
subject\ to \quad & g(\mathbf{u}) \leq 0
\end{aligned}
$$

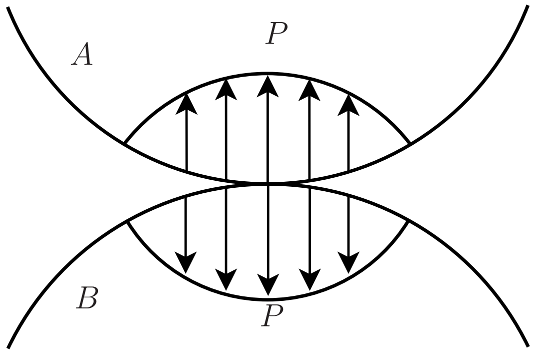

Figure 1: Physical problem.

$$
\begin{aligned}
min \quad & \Pi(\mathbf{u}) = \frac{1}{2}\mathbf{u}^t\mathbf{K}\mathbf{u} - \mathbf{f}^t\mathbf{u} \\
subject\ to \quad & g(\mathbf{u}) \leq TOLN
\end{aligned}
$$

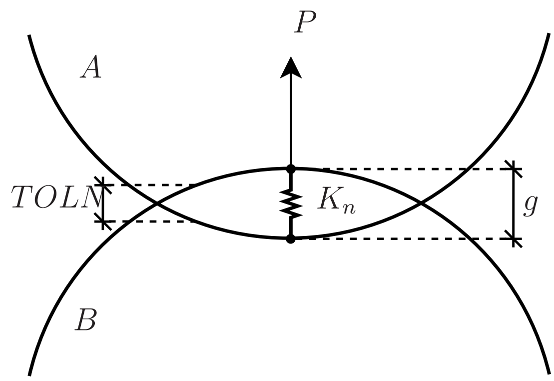

Figure 2: Numerical problem.

that represent contact pressure). These methods are usually implemented combined with numerical techniques like the Finite Element Method.

Finite element solvers

ANSYS [14] has available 3 different methods to solve the finite element analysis, and they are briefly described below:

- Sparse Direct: This method is based on factorising the system of equations into lower and superior matrices, reducing the computational cost by using specific techniques for sparse matrices. The advantage of this method is to be less sensitive to the conditioning of the stiffness matrix [14, 15].
- Preconditioned Conjugate Gradient - PCG: This is an iterative solver, where the minimisation problem is based on the Conjugate Gradient Method with a preconditioner step. Overall, the PCG requires less memory than the Spare Direct, hence being faster. It has been seen that contact analysis using either the Penalty or Augmented Lagrangian Method work well with this solver [15, 16].
- Frontal Direct: Here the global stiffness matrix is not assembled, so the system is solved one element at time. The method classifies the degrees of freedom of the system in dependent or independent, solving a triangular system for the independent degrees of freedom [17].

Normal contact stiffness

The normal contact stiffness is the most critical parameter in a contact analysis. It defines the amount of allowable penetration between the contact bodies. A small value can lead to a non suitable result, because of the large amount of penetration that will be present in the analysis. A high normal stiffness value could make the problem not to converge, since it will affect considerably the conditioning of the global stiffness matrix. It is expected to find a value large enough to have a small penetration, without affecting significantly the behaviour of the analysis.

Penetration limit

The penetration limit defines the acceptable penetration between contact elements. This parameter is applied to the normal direction of the contact surface and depends on the depth of the contact element. The contact element depth is stated as h in Fig. 3. The allowable penetration is the product of penetration limit factor (specified by the user) for the depth of the contact element. If the penetration is greater than this product, the solution is not considered converged, even if it satisfies other convergence criteria.

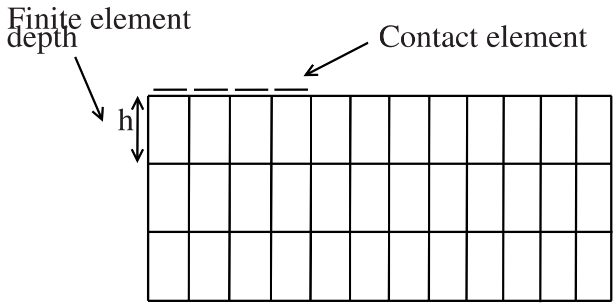

Figure 3: Definition of the element contact depth - From [13].

Contact algorithms

There are three main contact algorithms in ANSYS that will be described briefly:

- Penalty Method [2, 18].
- Lagrange Multipliers Method [15, 18].
- Augmented Lagrangian Method [1, 18].

Penalty Method

This method has the advantage that the non-penetration constraints are included directly in the objective function. The basis of the method is to penalise the non-feasible constraints with high values, and it is given by:

$$\begin{array}{cc} min & \Pi(\mathbf{u}) \\ subject\ to & g(\mathbf{u}) \leq 0 \end{array} \Rightarrow min \quad P(\mathbf{u}) = \Pi(\mathbf{u}) + \frac{r}{2} g(\mathbf{u})_+^2 \tag{3}$$

where $\Pi(\mathbf{u})$ is the potential energy function, $g(\mathbf{u})_+ = max[0, g(\mathbf{u})]$ is the non-penetration constraint and r is the penalty coefficient. The gradient is given by:

$$\nabla P(\mathbf{u}) = \mathbf{K}\mathbf{u} - \mathbf{f} + r g(\mathbf{u})_+ \nabla g(\mathbf{u}) \tag{4}$$

and the Hessian matrix is defined as:

$$H(\mathbf{u}) = \nabla^2 P(\mathbf{u}) = \mathbf{K} + r g(\mathbf{u})_+ [\nabla g(\mathbf{u})^t \nabla g(\mathbf{u}) + \nabla^2 g(\mathbf{u})] \tag{5}$$

It is clear that the penalty coefficient r affects the conditioning of the Hessian matrix, which corresponds to the stiffness matrix. For high values of r, the stiffness matrix can become ill-conditioned, and this is the main limitation of the Penalty Method [18].

The calculation of the contact pressure P in ANSYS is given by [14]:

$$P = \begin{cases} 0, & \text{if} \quad g(\mathbf{u}) > 0 \\ K_n g(\mathbf{u}), & \text{if} \quad g(\mathbf{u}) \leq 0 \end{cases} \tag{6}$$

where K_n is the normal contact stiffness and $g(\mathbf{u})$ is the penetration of the contact surfaces. The unit of K_n can vary in contact force models ($force/length$) or contact pressure models ($force/length^3$).

Lagrange Multipliers Method

The Lagrange Multipliers Method uses a dual formulation, where variables are added to the problem (Lagrange multipliers) and the problem is solved for these new variables. It is assumed that, if the Lagrange multipliers are known, the solution of the original constrained problem is easily calculated [18]. The Lagrangian function is stated as:

$$L(\mathbf{u}, \boldsymbol{\lambda}) = \Pi(\mathbf{u}) + \boldsymbol{\lambda}^t g(\mathbf{u}) \tag{7}$$

where $\boldsymbol{\lambda} = [\lambda_1 \ \lambda_2 \cdots \lambda_n]$ represents the Lagrange multipliers. The optimality conditions can be applied to the Lagrangian function allowing the solution for both displacements and Lagrange multipliers as a solution of the following matricial system of equations [15, 18]:

$$\begin{bmatrix} \mathbf{K} & \mathbf{B}^t \\ \mathbf{B} & \mathbf{0} \end{bmatrix} \begin{Bmatrix} \mathbf{u} \\ \boldsymbol{\lambda} \end{Bmatrix} = \begin{Bmatrix} \mathbf{f} \\ \boldsymbol{\alpha} \end{Bmatrix} \tag{8}$$

where the constraint conditions $g(\mathbf{u}) \leq 0$ can be written as $\mathbf{Bu} - \boldsymbol{\alpha} = \mathbf{0}$.

The implementation of this method in ANSYS consists in adding new variables to the system, instead of adding the stiffness of the contact elements directly into the global stiffness matrix [15]. The advantage of this approach is the possibility to enforce a zero penetration condition since the normal contact stiffness is not part of the method [19]. However, the additional Lagrange Multipliers require more iterations for the analysis to converge compared to other contact algorithms.

The contact pressures are calculated through the changes of the contact status, using the penetration limit ($FTOL$) and the maximum allowable tensile force ($TNOP$). The tensile force is the minimum force that acts in order to separate the contact bodies, and its default value in ANSYS is the unbalanced force convergence tolerance (difference between internal and external forces). The default value of the penetration limit is the displacement convergence tolerance. Table 1 shows how these two parameters influence the contact problem. Let g_{max} be the allowable penetration and $F_{i_{max}}$ the allowable tensile force, so that any value higher than these two parameters will result in a non representative solution.

Table 1: Description of the possible choices of the parameters g_{max} and $F_{i_{max}}$.

	$FTOL < g_{max}$	$FTOL > g_{max}$
$TNOP < F_{i_{max}}$	More iterations are needed to run the analysis, due to a greater number of changes in the contact status.	Penetration is overestimated, contact between two nodes may not be detected.
$TNOP > F_{i_{max}}$	Tensile force is overestimated, separation may not occur when necessary.	Both parameters are overestimated, the algorithm does not identify correctly the contact or the separation.

There are two potential limitations of employing the Lagrange Multiplier Method:

- There are excessively chattering (change of the contact status) in some nodes during the analysis, which could affect the solution precision and/or convergence;

- If the problem is overconstrained, i.e., there are additional constraints (as prescribed displacements) in the contact nodes, this contact algorithm could not converge or converge to an inaccurate result [14].

Augmented Lagrangian Method

This is a hybrid method of the Lagrange Multipliers Method and the Penalty Method. It represents competently the contact constraints using the penalty coefficient and the Lagrange multipliers, penalising the violations of the non-penetration constraints in the same manner of the Penalty Method. For the correct value of λ, the gradient of the Augmented Lagrangian function L_{aug} goes to zero, so the function is penalised exactly. Hence, it is not necessary to the penalty coefficient to be very high [18]. The Augmented Lagrangian function is defined as:

$$L_{aug} = \Pi(\mathbf{u}) + \boldsymbol{\lambda}^t g(\mathbf{u}) + \frac{1}{2} r g(\mathbf{u})_+^2 \tag{9}$$

In ANSYS, the contact pressure P is defined as:

$$P = \begin{cases} 0, & \text{if} \quad g(\mathbf{u}) > 0 \\ K_n g(\mathbf{u}) + \lambda_{i+1}, & \text{if} \quad g(\mathbf{u}) \leq 0 \end{cases} \tag{10}$$

The updated Lagrange multiplier λ_{i+1} is given by:

$$\lambda_{i+1} = \begin{cases} \lambda_i + K_n g(\mathbf{u}), & \text{if} \quad |g(\mathbf{u})| > TOLN \\ \lambda_i, & \text{if} \quad |g(\mathbf{u})| < TOLN \end{cases} \tag{11}$$

where λ_i is the Lagrange multiplier at iteration i.

Numerical examples

Contact between two cylinders

This example was first studied by [20] and is one of the few contact problems with analytical solution [21]. The problem is stated in Fig. 4.

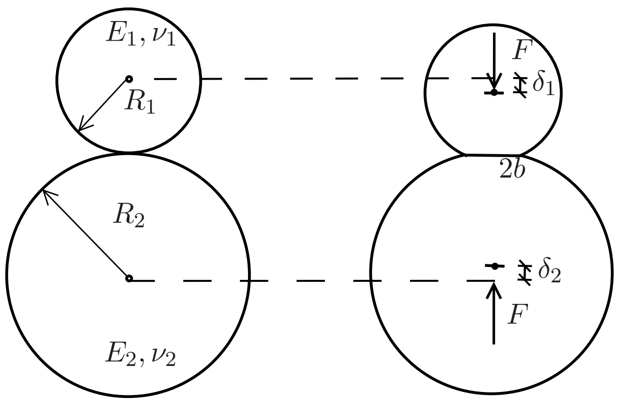

Figure 4: Contact between two cylinders.

Let $R_1 = 20$ mm and $R_2 = 10$ mm be the radii of cylinders with length $L = 1$ mm, $E_1 = E_2 =$ 210 GPa their Young's moduli, $\nu_1 = \nu_2 = 0.3$ their Poisson ratios, and an applied force $F = 1000$ N. The δ parameter indicates the total approximation of the cylinders, the b parameter represents the contact area and p_{max} is the maximum contact force.

The contact area b is defined as:

$$b = 1.13 \left(\frac{F}{L}(\eta_1 + \eta_2) \frac{R_1 R_2}{R_1 + R_2} \right)^{1/2} \tag{12}$$

where $\eta_1 = \frac{1-\nu_1^2}{E_1}$ and $\eta_2 = \frac{1-\nu_2^2}{E_2}$.

The total approximation distance $\delta = \delta_1 + \delta_2$ is defined by Eq. (13), where the particular case $\eta = \eta_1 = \eta_2$ was assumed:

$$\delta = 0.638 \frac{F}{L} \eta \left(\frac{2}{3} + ln\frac{2R_1}{b} + ln\frac{2R_2}{b} \right) \tag{13}$$

The maximum contact pressure is given by:

$$p_{max} = \frac{2F}{\pi b L} \tag{14}$$

Finally, the relative error is given by:

$$\text{Error}(\%) = \left(\frac{\text{current solution} - \text{analytical solution}}{\text{analytical solution}} \right) \times 100 \tag{15}$$

The ANSYS finite element mesh in the contact region used to analyse this problem is shown in Fig. 5 and the effects of the main parameters are in the following.

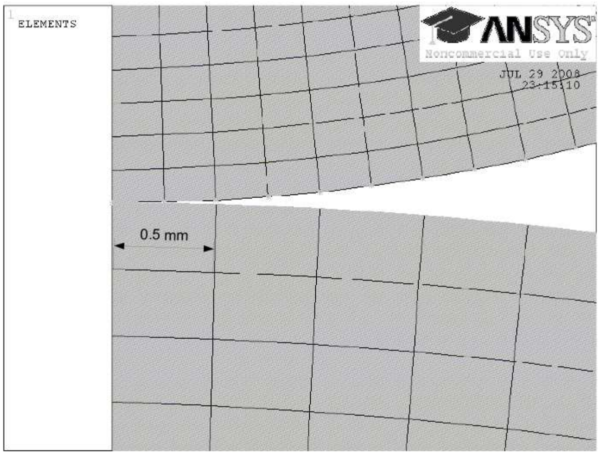

Figure 5: Refinement of the contact region - Contact between two cylinders.

Contact stiffness. Table 2 shows how the results vary with the variation of the normal contact stiffness when the other contact parameters (penetration limit, contact algorithm, solvers) remain the same. One can verify that for this example the variation is very small even for extremely high values of the stiffness factor. We chose $k_n = 5$ since in this case the error for the approximation is zero.

Table 2: Comparison between different contact stiffness - Contact between two cylinders.

	Analytic	$k_n = 1$	Error(%)	$k_n = 5$	Error(%)	$k_n = 10$	Error(%)	$k_n = 100$	Error(%)
δ	0.0275	0.0277	0.73	0.0275	0.0	0.0274	-0.36	0.0274	-0.36
b	0.2716	0.2618	-3.62	0.2618	-3.62	0.2618	-3.62	0.2618	-3.62
p_{max}	2343.8	2353.8	0.43	2488.8	6.18	2508.1	7.01	2526.1	7.78

Table 3: Comparison between different penetration limit factors - Contact between two cylinders.

$FTOL$	Maximum penetration	Maximum pressure	Number of iterations
1	1.8436×10^{-4}	2488.8	3
0.1	1.8436×10^{-4}	2488.8	3
0.01	1.8437×10^{-4}	2488.9	5
0.001	9.4089×10^{-5}	2514.9	30
0.0001	1.3536×10^{-5}	2527.4	35

Penetration limit. The influence of the variation of the penetration limit are given in Table 3. When the penetration limit factor $FTOL$ decreases, there is a reduction of the penetration, and an increase in the number of iterations used to find the solution. In this case, the best choice is $FTOLN = 0.1$, since penetration is sufficiently small.

Contact algorithms. The results obtained with the Penalty Method, Lagrange Multipliers Method and Augmented Lagrangian Method are presented in Table 4. The results of the Penalty Method and the Augmented Lagrangian Method are the same in terms of penetration and contact pressure. One possible reason for this is that this example is very well behaved, so that the three solution methods work well.

Table 4: Comparison between different contact algorithms - Contact between two cylinders.

Contact algorithm	Maximum penetration	Maximum pressure	Number of iterations	δ	b
Penalty	8.7183×10^{-4}	2488.8	3	0.0275	0.2618
Augmented Lagrangian	8.7183×10^{-4}	2488.8	3	0.0275	0.2618
Lagrange Multipliers	4.1691×10^{-6}	2529.1	2	0.0274	0.2618

Solvers. Here the different finite element solvers are compared and the results are shown in Table 5. The results in terms of penetration level, contact pressure, approximation and contact area are the same for every method, the only difference lying in the computational time, which is very small for each method.

Table 5: Comparison between different solvers - Contact between two cylinders.

Contact algorithm	Maximum penetration	Maximum pressure	Time (CPU)	δ	b
Sparse	1.2252×10^{-5}	2526.8	7	0.0274	0.2618
PCG	1.2252×10^{-5}	2526.8	8	0.0274	0.2618
Frontal	1.2252×10^{-5}	2526.8	9	0.0274	0.2618

Automotive connecting rod

This is a classical example of contact in mechanical systems and has been extensively studied [12, 13, 22], just to cite a few references. Fig. 6 and 7 represent only a quarter of the connecting rod and pin geometry and some of the measures of the model, respectively.

The finite element mesh has 63702 nodes and 25165 elements. In this example, two types of elements were used: tetrahedral elements with 10 nodes for the connecting rod mesh and hexahedral elements with 20 nodes for the pin mesh.

A clearance of 18 μm between the connecting rod and pin is assumed. The material properties are $E = 210$ GPa and $\nu = 0.3$. The loading is $F = 40000$ N distributed evenly across the nodes as indicated in Fig. 7.

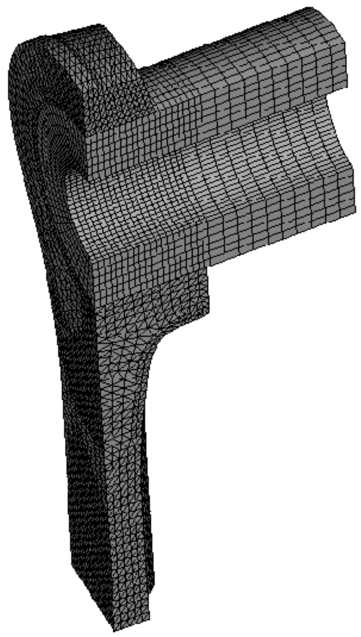

Figure 6: Finite element mesh - Automotive connecting rod - From [13].

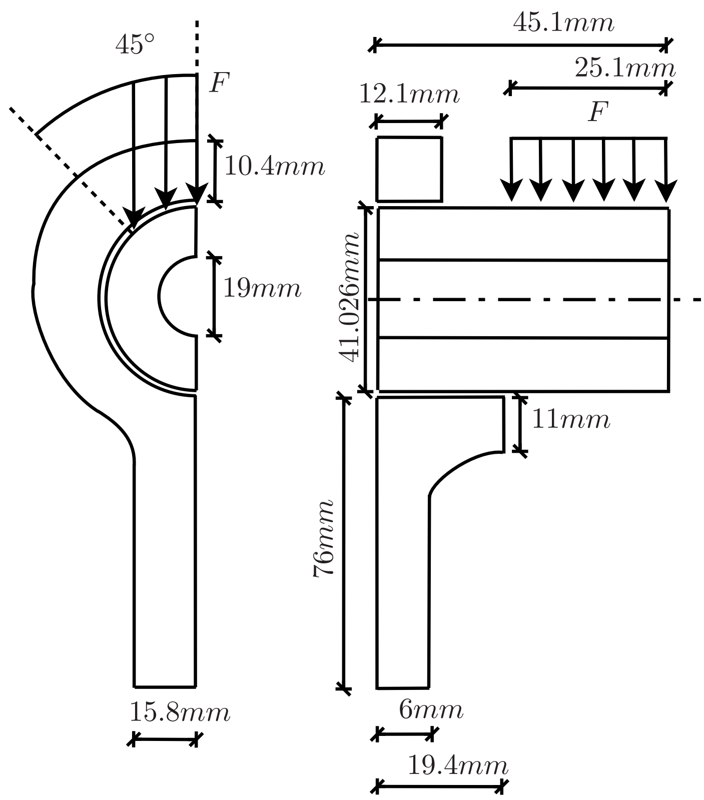

Figure 7: Simplified geometrical model - Automotive connecting rod - From [13].

Contact stiffness. For large contact problems, the conditioning of the stiffness matrix has to be taken into account. Fig. 8 illustrates the relation between the penetration and the condition number of the stiffness matrix in terms of the contact stiffness factor k_n.

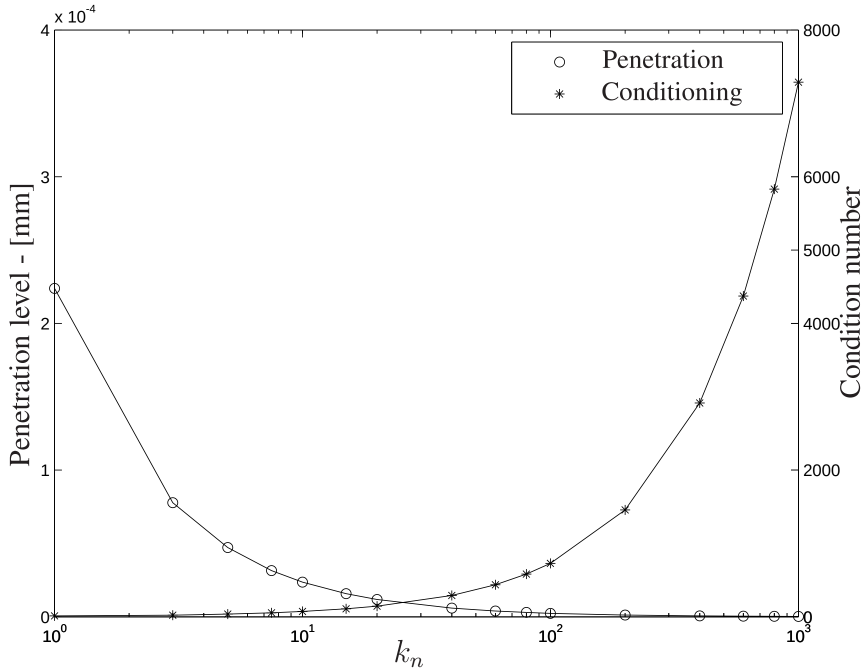

Figure 8: Penetration and conditioning of the stiffness matrix - Automotive connecting rod.

High values of the normal contact stiffness lead to low penetration levels but can also lead to a high condition number. In this case, when $k_n = 25$ the condition number is not too elevated and the penetration level is kept low.

Table 6 shows the results for different stiffness factors in terms of the number of iterations and computational time. As expected, high values of k_n require larger number of iterations for the analysis to converge, as well as more processing time.

Table 6: Contact stiffness factor and the condition number, iterations and computational time - Automotive connecting rod.

k_n	Condition number	Number of iterations	Time (CPU)
1	12.73	10	431
3	21.87	10	425
5	36.45	12	516
7.5	54.67	13	582
10	72.90	15	717
15	109.35	16	896
20	145.79	15	896
40	291.59	21	1589
60	437.38	20	1810
80	583.18	26	2632
100	728.97	37	4082
200	1457.95	28	4120
400	2915.90	33	5510
600	4373.85	37	7174
800	5831.80	36	7889
1000	7289.75	43	18465

Fig. 9 illustrates the maximum penetration between the contact surfaces and the maximum contact pressure in terms of the contact stiffness factor. One can observe that P_{max} increases along with the contact stiffness until a certain threshold, when it does not vary even if we keep increasing the value of k_n. In this example this threshold is attained for $k_n = 10$, and will be used further in this paper.

Penetration limit. Table 7 shows the relation of the maximum penetration level between contact bodies in terms of the contact stiffness factor and the penetration limit factor.

Table 7: Penetration in terms of penetration limit and contact stiffness factors - Automotive connecting rod.

$FTOLN$	k_n	Max. penetration level	Max. pressure	Iterations
0.1	10	2.3606×10^{-5}	414.85	15
0.01	10	2.3606×10^{-5}	414.85	15
0.001	10	2.3606×10^{-5}	414.85	15
10^{-4}	10	2.3606×10^{-5}	414.85	17
10^{-5}	10	–	–	no convergence
0.1	0.1	2.0729×10^{-3}	364.27	7
0.01	0.1	2.0729×10^{-3}	364.27	7
0.001	0.1	1.2447×10^{-3}	398.76	7
10^{-4}	0.1	1.2413×10^{-4}	409.59	23
10^{-4}	1	8.6332×10^{-5}	415.53	12

First, we change the penetration limit factor but maintain fixed the contact stiffness factor in $k_n =$

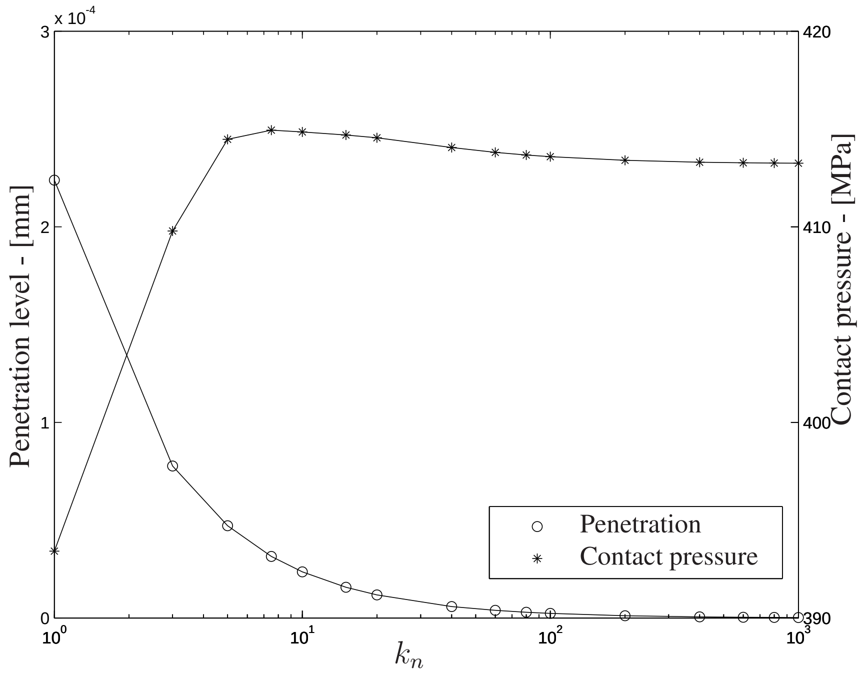

Figure 9: Penetration and contact pressure - Automotive connecting rod.

10. The maximum penetration level remains constant for these configurations. Reducing the contact stiffness factor to $k_n = 0.1$, we note changes of the penetration level for different penetration limits. The best results in terms of number of iterations and penetration level is obtained when $k_n = 1$ and $FTOLN = 10^{-4}$. On the one hand, when the contact stiffness assumes high values, the penetration limit may not be used, so the Lagrange multipliers are not updated and the Augmented Lagrangian Method is reduced to the Penalty Method. On the other hand, if the contact stiffness value is too underestimated, the combination of low contact stiffness and low penetration limits will require more iterations so the contact analysis will converge.

Contact algorithms. Some results for different contact algorithms are shown in Table 8, where the penetration limit factor is fixed to $FTOLN = 10^{-4}$ and the contact stiffness factor is assumed to be $k_n = 1$ or $k_n = 10$.

Table 8: Contact algorithms - Automotive connecting rod.

Contact algorithm	Contact stiffness	Maximum penetration	Maximum pressure	Maximum stress	Number of iterations	Time (CPU)
Penalty	10	2.3606×10^{-5}	414.85	652.39	15	713
Augmented Lagrangian	10	2.3606×10^{-5}	414.85	652.39	17	803
Penalty	1	2.2387×10^{-4}	393.42	638.13	10	430
Augmented Lagrangian	1	8.6332×10^{-5}	415.53	647.79	12	505
Lagrange Multipliers	–	1.7247×10^{-10}	494.2	769.44	11	2888

For $k_n = 10$, the same results were obtained for both Penalty and Augmented Lagrangian Method. As mentioned earlier, for high values of the normal contact stiffness, the Augmented Lagrangian behaves as the Penalty Method.

For $k_n = 1$, we observe differences between the Penalty and Augmented Lagrangian Method. The penetration level is lower when using the Augmented Lagrangian Method due to the Lagrange multipliers which were updated for this configuration. The difference between the contact pressures is -4.80% of the Penalty Method assuming the reference is $k_n = 10$. For the Augmented Lagrangian

Method, the error is 0.55%.

The Lagrange Multipliers Method presented the lowest penetration level, however it required over the double of the computational time of the other contact algorithms. Moreover, the contact pressure was 19% over the reference of $k_n = 10$. The probable cause of this variation is the excessive chattering of contact status during the analysis.

Solvers. Table 9 presents the results for different finite element solvers for $k_n = 10$ and $FTOLN = 0.1$. Again, the penetration level and contact pressure are the same but with distinct computational time for every solver.

Table 9: Comparison between different solvers - Automotive connecting rod.

Contact algorithm	Maximum penetration	Maximum pressure	Number of iterations	Time (CPU)
Sparse	2.3606×10^{-5}	414.85	15	1836
PCG	2.3606×10^{-5}	414.85	15	718
Frontal	2.3606×10^{-5}	414.85	15	19145

Fig. 10 illustrates how the computational time varies with the contact stiffness factor for the 3 considered solvers. For $k_n \geq 150$, the Sparse solver finishes the analysis faster than the PCG. When $k_n < 150$, the PCG is the fastest solver. The computational time required by the Frontal solver is very elevated even for low values of the contact stiffness factor. The reason for this behaviour is the architecture of the solver in terms of memory allocation [14].

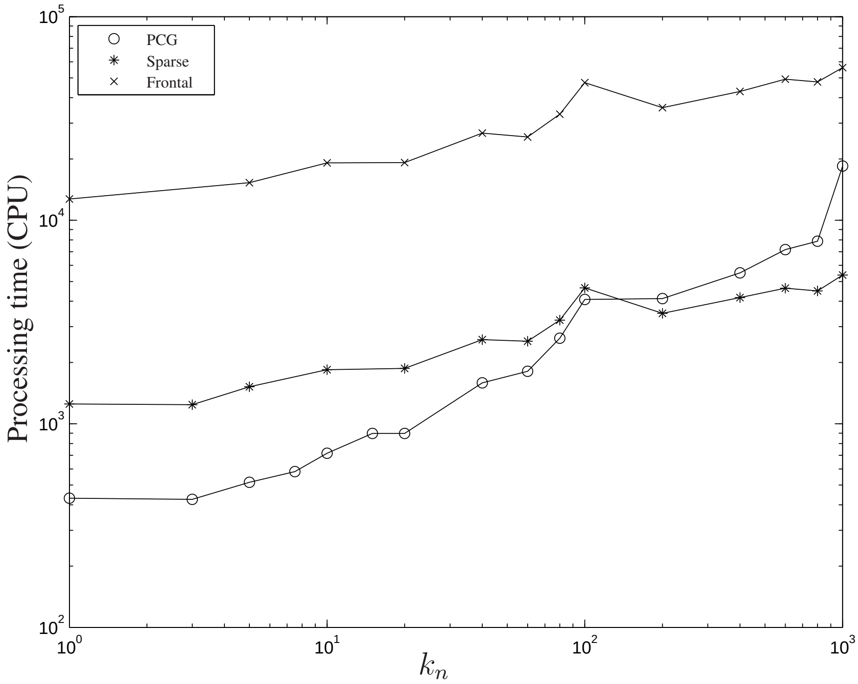

Figure 10: Computational time of solvers in terms of the contact stiffness factor - Automotive connecting rod.

Conclusions

In this paper, the main contact parameters of the commercial finite element analysis software ANSYS have been studied. The main contact parameters are the normal contact stiffness, the penetration limit and the contact algorithms. There are other contact parameters, but their influence is not as critical as the aforementioned parameters. The normal stiffness and the penetration limit are responsible for regulating the level of acceptable penetration between the bodies in contact. If the normal stiffness assumes values too elevated, the penetration limit may not be considered, and the Augmented Lagrangian Method will behave as the Penalty Method. Moreover, the stiffness matrix can become ill-conditioned, affecting the convergence and also leading to higher amounts of computational time. The ideal situation is to obtain a compromise between the normal contact stiffness and the penetration limit. The Lagrange Multiplier can provide penetration levels close to zero since its formulation does not use the contact stiffness. However, the method is more sensible to instabilities during the contact analysis, which can point to non suitable solutions. The influence of friction in contact problems can be investigated in future works.

Acknowledgements

The first author acknowledges the Faculty of Science, Durham University, for his Postdoctoral Research Associate funding. The authors also thank *ThyssenKrupp Metalúrgica Campo Limpo*, Brazil for the financial support. Figs. 3, 6 and 7 have been reproduced from reference [13] with permission from Elsevier.

References

[1] J. C. Simo, T. A. Laursen, An augmented Lagrangian treatment of contact problems involving friction, Comput. and Struct. 42 (1992) 97-116.

[2] D. Peric, D. R. J. Owen. Computational model for 3-D contact problems with friction based on the penalty method, Int. J. Numer. Meth. Eng. 35 (1992) 1289–1309.

[3] M. A. Puso, T. A. Laursen, A mortar segment-to-segment contact method for large deformation solid mechanics, Comput. Meth. Appl. Mech. Eng. 193 (2004) 601-629.

[4] A. P. C. Dias, A. L. Serpa, M. L. Bittencourt, High-order mortar-based element applied to non-linear analysis of structural contact mechanics. Comput. Meth. Appl. Mech. Eng. 294 (2015) 19-55.

[5] E. A. Fancello, Formulação variacional do problema de contato com atrito: resolução via regularização, Master's thesis, COPPE - UFRJ, Rio de Janeiro, Brazil, 1989.

[6] H. J. C. Barbosa, Algoritmos numéricos para problemas de contato em elasticidade, PhD thesis, COPPE - UFRJ, Rio de Janeiro, Brazil, 1986.

[7] T. A. Laursen, Computational Contact and Impact Mechanics, 2nd ed., Springer, 2003.

[8] A. L. Serpa, F. Iguti, Contact with friction using the augmented Lagrangian method: a conditional constrained minimization problem, J. Bra. Soc. Mech. Sci. 22 (2000) 273-289.

[9] A. R. Mijar, J. S. Arora, An augmented Lagrangian optimization method for contact analysis problems, Struct. Multidiscip. Optim. 28 (2004) 99-112.

[10] G. Stadler, Path-following and augmented Lagrangian methods for contact problems in linear elasticity, J. Comput. Appl. Math. 203(2007) 533-547.

[11] M. Ciavarella, A. Baldini, J. R. Barber, A. Strozzi, Reduced dependence on loading parameters in almost conforming contacts, Int. J. Mech. Sci. 48 (2006) 917-925.

[12] A. Strozzi, F. De Bona, Hoop stresses in the con-rod small end, Proced. Int. Mech. Eng. Part D: Automob. Eng. 219 (2005) 1331-1345.

[13] G. Hattori, A. L. Serpa, Contact stiffness estimation in ANSYS using simplified models and artificial neural networks, Finite Elem. Anal. Des. 97 (2015) 43-53.

[14] ANSYS, ANSYS Contact Technology Guide, 2005.

[15] K.-J. Bathe, Finite Element Procedures in Engineering Analysis, Prentice Hall, 1996.

[16] J. Dunagan, N. J. A. Harvey, Iteratively constructing preconditioners via the conjugate gradient method, Proced. 39th annu. ACM Sym. Theor. Comput. (2007) 207-216.

[17] B. M. Irons, A frontal solution program for finite elements analysis, Int. J. Numer. Meth. Eng. 2 (1970) 5-32.

[18] D. G. Luenberger, Linear and Nonlinear Programming, 2nd ed., Addison-Wesley Publishing Company, 1989.

[19] D. Peyrou, F. Coccetti, F. Pennec, H. Achkar, P. Pons, R. Plana, A new methodology for RF MEMS contact simulation 9th Int. Conf. Therm. Mech. Multi-Phys. Simulat. Exp. Microelectron. Microsyst. (2008) 1-4.

[20] H. Hertz, On the contact on elastic solids, J. Math. 92 (1881) 156-171.

[21] N. Chandrasekaran, W. E. Haisler, R. E. Goforth, Finite element analysis of Hertz contact problem with friction, Finite Elem. Anal. Des. 3 (1987) 39-56.

[22] P. S. Shenoy, A. Fatemi, Dynamic analysis of loads and stresses in connecting rods, Proced. Int. Mech. Eng. Part C: Mech. Eng. Sci. 220 (2006) 615-624.