PEDAGOGY OF THE CONCEPT OF LOAD AT A DISTANCE IN F.E.A. CODES (BASIC CASE STUDIES)

Nader G. Zamani(*)
Department of Mechanical, Automotive and Materials Engineering (MAME), University of Windsor, Windsor, Ontario, Canada
(*) Email: zamani@uwindsor.ca

ABSTRACT
This work is an attempt to provide an elementary account of explaining the rationale and usage of the “rigid” elements available in many commercial F.E.A. software. Such capabilities were originally introduced half a century ago in the early commercial programs. This may partly be the reason that after so long, it is one of the least understood and most widely abused elements in a commercial code, particularly by the engineering undergraduates and industrial designers. In the current presentation, a several case studies are introduced which is of pedagogical value for educational purposes and hopefully will shed some light on this issue.

Keywords: finite elements, rigid elements, multipoint constraint, virtual part, pedagogy.

INTRODUCTION
One of the earliest finite element packages, the NASTRAN program was developed by NASA in 1960s. This software is still around and after substantial modifications, is the backbone of several commercially available FEA packages (MSC, 2015). The computer hardware limitations in that era, necessitated the usage of special type elements (novel elements) that were dramatically cutting the cost of computing. One such particular capability was known as “Rigid” element, which is still available in just about any commercial FEA package (Zamani, 2016).

The rigid elements, depending upon the software, are referred to by different names. It is widely known as RBE2 and RBE3. Since these are not truly elements in the conventional sense, in some software such as CATIA v5 they are referred to as “Virtual” elements. As indicated in the abstract, these are probably the most widely misunderstood concepts in FEA. Part of the reason may be that they are not completely documented in the software manuals. Particularly, there is a lack of good and understandable examples where the elements are explained in detail and their results are interpreted satisfactorily.

One of the main objectives of this expository paper is to discuss such element in terms of scenarios where an average user can see and appreciate. To put things in perspective, the standard example that is presented in just about any commercial software is illustrated below.

A block is fixed at one end and a remote point is located a certain distance away from the face. An axial force applied at the remote point is compressing the block (only symmetric compression), see Fig.1 for the problem statement. The results of these problems using RBE2 and RBE3 are displayed in Fig. 2 and Fig. 3 respectively.
Notice that under RBE2, the compressed face of the block remains planar whereas, with RBE3, there is a deformation of the face with severe stresses at the corner locations of the face. The question often asked by students is, “so which one do I use, and how do I know that these results are correct?”

Fig. 1 - Remote handler point “H” carries the load of 100kN

Fig. 2 - von Mises and displacement contour plots using the “RBE2” elements

Fig. 3 - von Mises and displacement contour plots using the “RBE3” elements

Another classical example is the part shown in Figure 4. Ignoring the dimensions and the actual material properties, one end of the block is clamped and the hole is loaded through the handler point H. The term “Handler” and “Remote” points are used interchangeably.
The deformation of the above part is shown in Fig. 5. Note that in the case of “RBE2” element, the circular hole is undeformed, whereas there is clear deformation when “RBE3” elements are employed.

**VIRTUAL ELEMENTS IN CATIA V5**

Since the CATIA v5 software will be used in this paper as our mail tool, the terminology used in the software need to be described. In CATIA, such elements are known as “Virtual Parts”, or “Virtual Elements”. There are five such elements but only four will be addressed in the present paper. The toolbar which carries such entities and the corresponding icons are shown in Fig. 6. These elements will be described in terms of a bar under different loading conditions.
Consider the block shown in Fig.7 whose left end is clamped and the right end is loaded to cause bending. The block is made of a non-homogeneous material such that a certain length of it is much more rigid that the other piece. In Fig. 7, as indicated, the portion adjacent to the clamp end is flexible (ie deformable) and the right portion moves as a rigid body. This being the case, one can mesh the deformable portion, and model the right piece as a rigid body. That is precisely what a “Rigid”, or “Smooth” virtual part does. The only difference between the two, is that interface of the two portions is assumed to be non-deformable if a “Rigid” virtual part is used.

In the case of a “Smooth” virtual part, interface deforms and does not remain a flat surface.

The problem shown in Fig.8 is exactly the same as the one described above. The only difference is that although the right side (the part adjacent to the load) will not be meshed with solid elements, the stiffness of this portion is provided to the software as an input.

Fundamentally, there is no difference between the “Rigid Spring” and “Smooth Spring” virtual parts except for the deformability of the interface. The dialogue box which allows you to provide this data in the Catia v5 software is displayed in Fig.9.
As it can be seen, the stiffnesses in all six directions (three translations, and three rotations) can be assigned. These are not ad hoc numbers but some hand calculations can/must be performed to estimate the values. This type of calculation will be presented at a later point in the paper.

![Rigid Spring Virtual Part](image)

**Fig. 9 - Dialogue box for Catia v5**

**RESULTS**

In this section, six case studies will be discussed to demonstrate the ideas presented in the introduction. However, in view of the fact that there is no fundamental difference between “Rigid” and “Smooth” virtual parts, the discussion will be limited to “Rigid” virtual parts. The material is assumed to be linear elastic with Young’s modulus of 200 GPa and Poisson’s ratio of 0.3.

In all these case studies, linear 4-noded tetrahedron elements are used and to alleviate the concern about the mesh size, a specified value of 1mm is used for mesh generation. This will result in a very fine mesh throughout the part.

**CASE STUDY 1, AXIALLY LOADED BAR, RIGID VIRTUAL PART:**

The bar of dimensions 10x10x150 mm is clamped at its left end point. The last 50 mm of the bar is assumed to considerably more rigid than the remaining portion. This is why that section of the bar is modelled as a rigid virtual part. The bar is loaded axially at the handler point resulting in a compression of 0.1 mm as shown in Fig.10. The free body diagram of both sections are displayed in Fig.11.
The bar deflection and the axial principle stress contours for the meshed portion of the structure is given in Fig. 12. The enforced deflection of 0.1 mm requires a load of $F = -AE \frac{\delta}{L}$. Here, 

$\delta = 0.1\text{mm}$ and $L = 100\text{mm}$. Therefore, the calculated axial principle stress is $\sigma = \frac{F}{A} = 200\text{ MPa}$. To have a better idea of what the predicted value in the middle and the end sections of the bar are, three “Groups” in Catia are created. This is shown in Fig. 13. This allows us to plot the stress distribution only in these section instead of the entire bar.

![Fig. 10 - Original part, modeled part, meshed part](image)

![Fig. 11 - The Free body diagram of the left side of bar meshed](image)

![Fig. 12 - The axial principle stress and the deflection](image)
The principle stress in these sections are plotted in Fig. 14. In the middle section, the predicted value is 201 MPa (compressive) as expected. One does not expect to get an agreement at the loaded end. The principle stress in the vicinity of the clamp is not far from 200 MPa.

Although from the engineering point of view, the stress distribution at the support may be acceptable, the reason behind the bizarre non-uniform contour is because of the clamped condition. We have also made a run with roller support at the wall and generated the stress distribution which is displayed in Fig. 15. Note that the contour shows a compressive principle stress of -201 MPa but is more uniform.
One should be careful about the “Group” at the support. We should avoid including the vertices where the zero displacement (to prevent rigid body motion) are specified. There are artificial stresses developed due to the zero displacements. The selected group is shown in Fig. 16 for the sake of completeness.

**Fig. 16 - The group at the roller support must avoid the vertices with specified zero displacements**

**CASE STUDY II, AXIALLY LOADED BAR, RIGID SPRING VIRTUAL PART:**

This is exactly the same problem as in the Case Study I except that the stiffness of the right side of the bar is also take into account. The situation is summarized in Fig. 17. The 150 mm bar is shown but it is felt that there is no reason to model the end 50 mm. However, this is not to say that this 50 mm section is axially rigid. Therefore, one can model this section with “Rigid Spring” virtual part. The axial stiffness of the spring has to be calculated realistically based on dimensions and the material properties. This calculation will be making references to Fig. 18.

**Fig. 17 - The use of a “Rigid Spring” virtual part**

In order to assess the quality of the finite element simulations, some hand calculations is presented below.

The free body diagram (FBD) of the two portions of the bar is shown in Fig.18. The elementary strength of materials formula, estimates the axial stiffness of the 50 mm section as
\[ k_2 = \frac{AE}{d} = \frac{AE}{0.05} = 4 \times 10^8 \text{ N/m}. \] This will be the “Translational” spring constant that will be specified in the dialogue box shown in Fig. 9. Naturally the proper direction must be used.

To find a rough first order solution to the original problem, the axial stiffness of the 100 mm portion of the bar is calculated from \[ k_1 = \frac{AE}{L} = \frac{AE}{0.1} = 2 \times 10^8 \text{ N/m}. \] The two portions of the bar can be viewed as two springs in series with the equivalent stiffness of \[ k_{eq} = \frac{k_1k_2}{k_1+k_2} = \frac{4}{3} \times 10^8 \text{ N/m}. \]

The force necessary to cause a 0.1 mm displacement at the handler point is \[ F = k_{eq}\delta = \frac{4}{3} \times 10^8 \times 0.0001 = 13333 \text{ N}. \]

The axial stress in the bar is calculated from \[ \sigma = \frac{F}{A} = \frac{13333}{0.0001} = 133 \times 10^6 \text{ Pa} = 133 \text{ MPa}. \]

The tip deflect of the modelled section of the bar is calculated from \[ u = \frac{F}{k_{eq}} = 67 \times 10^{-6} \text{ m} = 0.067 \text{ mm}. \]

The finite element results with a clamped end and “Rigid Spring” virtual part is Fig. 19. As far as the displacement plot is concerned, it predicts the exact value as predicted by the hand calculations. As far as the axial principle stress is concerned, it is definitely constant and the color range is in agreement with the 133MPa predicted above. To create a more informative plot, the stress distribution in the constructed groups are presented in Fig. 20. Now, one can clearly see that the stress distribution is predicted to be 134 MPa.
There are a few comments in order. Based on the information presented in the Case Study I, if the clamped end is replaced with a roller support, the stress distribution at the clamped end will be more uniform and very close to the theoretical value of 133 MPa.

The results predicted by “Smooth Spring” virtual part is essentially identical to the calculation above. The difference will be the deformability of the interface. In view of this observation, we choose to skip this element.

**CASE STUDY III, TORSIONALY LOADED BAR, RIGID VIRTUAL PART**

The circular bar of radius $r = 10$ mm and length 150 mm is clamped at its left end point and subject to a twist angle of 0.1 degree at the other end. This is familiar torsional loading on a circular bar. The situation is depicted in Fig. 21. The last 50 mm of the bar is assumed to considerably more rigid than the remaining portion. This is why that portion of the bar is modelled as a rigid virtual part.

For the reader to have an appreciation of the fineness of the mesh, the zoomed in view of the clamped end is shown in Fig. 22. Keep in mind that the radius of the bar is 10 mm and the element size is roughly 1 mm. The elements are of linear tetrahedron type.

The free body diagram of the two sections are shown in Fig. 23. Due to the fact that the right side is modeled as being rigid, the left side also twists by 0.1 degree. It is obvious that the torque applied to the left side has the same value as $T$. 

---

For the reader to have an appreciation of the fineness of the mesh, the zoomed in view of the clamped end is shown in Fig. 22. Keep in mind that the radius of the bar is 10 mm and the element size is roughly 1 mm. The elements are of linear tetrahedron type.
The torsional principle stress contours for the meshed portion of the structure (generated by FEA) is given in Fig. 24. The hand calculation below, using the torsion formulas will be used to assess the results.

The twist of $\theta = 0.1^\circ$ requires a torque of $T = \frac{GL\theta}{J}$. Here $G = \frac{E}{2(1+v)}$ is the shear modulus and $J$ is the polar moment of inertia of the bar. For the left end of the bar which is modeled, $L = 100mm$. Therefore, the calculated torsional principle stress is $\tau = \frac{T\theta}{J} = 13.47 \times 10^6 Pa = 13.47 MPa$. To have a better idea of what the predicted value in the middle and the end sections of the bar are, they are also plotted in Fig. 25, using the groups created earlier.
Notice that the contour plot in Fig. 25 for the middle section is in excellent agreement with the hand calculation value of 13.47 MPa. The results of the “Smooth” virtual part will be so close to the “Rigid” virtual part that will not be presented here. As pointed out earlier, the difference is in the deformation of the interface.

**CASE STUDY IV, TORSIONALLY LOADED BAR, RIGID SPRING VIRTUAL PART**

This is the same type of problem as in the Case Study III except that the torsional stiffness of the right side of the bar is also taken into account. The situation is summarized in Fig. 26. The 150 mm bar is shown but it is felt that there is no reason to model the end 50 mm. However, this is not to say that this 50 mm section is torsionally rigid. Therefore, one can model it with “Rigid Spring” virtual part. The torsional stiffness of the spring has to be calculated realistically based on dimensions and the material properties.

![Fig. 26 - Original part, modeled part, meshed part](image)

In order to assess the quality of the FEA predictions, some hand calculation based on strength of materials formulas are made. These are in reference to the FBD displayed in Fig. 27. The 50 mm segment has a torsional stiffness given by $k_2 = \frac{GL}{d} = \frac{GJ}{0.05} = 24166$ Nm/rad. This will be the “Rotational Stiffness” spring constant that will be provided to the software, see Fig. 9.

The left side of the bar which meshed with elements, has a torsional stiffness, given by $k_1 = \frac{GJ}{L} = \frac{GJ}{0.1} = 12083$ Nm/rad. The two torsional spring are acting in series with an equivalent stiffness of $k_{eq} = \frac{k_1k_2}{k_1+k_2} = 8055$ Nm/rad.
This equivalent stiffness allows us to calculate the torque necessary to cause a $0.1^\circ$ twist angle at the handler point. Therefore, $T = k_{eq}\theta = 8055 \times \frac{0.1 \times (2\pi)}{360} \approx 14$ Nm.

The principle torsional stiffness on the surface of the bar can then be calculated from the following expression. $\tau = \frac{T_r}{J} = \frac{14 \times 0.01}{J} = 8.095 \times 10^6 Pa = 8.095 MPa$.

The finite element calculation are compared to this reference value. The torsional principle stress for the entire structure and the three sections (clamped end, middle, and loaded sections) are given in Fig. 28 and 29 and are very satisfactory. It is important to keep in mind in a torsion problem, the stress (torsional stress) varies linearly with distance to the axis of the bar. Therefore, the maximum stress is assumed on the surface of the bar which is farthest away from the axis. The linear variation can easily be confirmed by the uniform concentric circles in the contours within the structure.

Fig. 28 - The principle torsional stress distribution for the entire model

Fig. 29 - The principle torsional stress in three regions along the bar

The agreement of the stress distribution in the middle section of Fig. 29 is excellent. The results of the “Smooth” virtual part will be so close to the “Rigid” virtual part that will not be presented here.

Furthermore, a roller support at the clamp would have produced a more uniform distribution at the restrained end.

**CASE STUDY V, BAR UNDER BENDING, RIGID VIRTUAL PART**

The 150 mm bar shown in Fig. 30 is clamped at the left end and subjected to a downward load of 1000N. It is assumed that the 50 mm section on the right side is so much stiffer than the rest of the beam that it does not have to modelled. The cross section of the beam and its material properties are the same as in Case I. The last 50 mm of the bar will not be meshed and instead modeled as a “Rigid” virtual part.
Contrary to the previous four case studies where the finite element results were presented first and then the theoretical solution, we tend to reverse the order here. This decision is mainly because of the more complex nature of the problem under consideration. The free body diagram of the two sections of the beam is displayed in Fig. 31. Concentrating on the left section of the bar which will be meshed, it is subjected to a downward load of “F” and a bending moment of “M”. This moment is the result of the couple “Fd” which has been created due to removal of the rigid section.

The basic strength of materials formulas needed to arrive at a reference value for comparison purposes is given in Fig. 32. On the left side, you see the cantilever beam subject to two separate loads. The vertical downward force F and the bending moment M. On the right side, the superposition of the two loads are given. This is precisely the scenario in the “Rigid” virtual part problem as displayed in Fig. 31.
For the present problem, the following data is used.

- \( b = h = 0.01 \, \text{m} \) , ie square cross section. \( \rightarrow I = \frac{1}{12} b h^3 = 8.333 \times 10^{-10} \, \text{m}^4 \)
- \( L = 0.1 \, \text{m} \) , the length of the modeled section of the beam
- \( d = 0.05 \, \text{m} \) , the length of the “Rigid” section
- \( F = 1000 \, \text{N} \) , the applied load
- \( E = 200 \, \text{GPa} \) , Young’s modulus
- \( \nu = 0.3 \) , Poisson’s ratio

The above data will give the following calculated reference values.

- \( \delta = \frac{FL^3}{3EI} = 3.5 \times 10^{-3} \, \text{m} = 3.5 \, \text{mm} \), tip deflection of the modeled section of the beam.
- \( \sigma = \frac{M_{\text{wall}} c}{l} = \frac{F(l+d)^2}{l} = 900 \times 10^6 \, \text{Pa} = 900 \, \text{MPa} \), bending stress at the wall

As for the deflection of the handler point, ie the remote location of the applied load, we refer to figure 33. And the following value is obtained.

\[
\delta_{\text{Handler point}} = \frac{FL^3}{3EI} + \frac{ML^2}{2EI} + \theta d = \frac{FL^3}{3EI} + \frac{(Fd)L^2}{2EI} = \frac{FL^3}{3EI} + \frac{(Fd)L^2}{2EI} + \left[ \frac{FL^2}{3EI} + \frac{(Fd)L}{EI} \right] d = \\
= 6.5 \times 10^{-3} \, \text{m} = 6.5 \, \text{mm}
\]
Fig. 33 - The treatment of the displacement of the Handler point

\[ \delta_{\text{Handler}} = \delta + \theta d = \delta + \dot{\delta} + \left( \ddot{\theta} + \ddot{\theta} \right) d = \]

\[ \frac{FL^3}{3EI} + \frac{ML^2}{2EI} + \left[ \frac{FL^2}{2EI} + \frac{ML}{EI} \right] d \]

We are now in a position to present the FEA results. The deflections of the modeled section and the “Rigid” virtual part are displayed in Fig. 34. There is a good agreement between the contour plot and the theoretical calculations above.

We have also replaced the clamped end with the roller end condition shown in Fig. 36. Note that to avoid the rigid body motion, one also has to fix two of the vertices. This needs to be done carefully so that no fictitious (artificial) stresses are developed at these vertices. This is sometimes referred to as the “123” rule and is very common in “Inertia Relief” modelling.
The contours of the deflection and the principle bending stresses are shown in figures 37, 38, and 39 respectively. There are very minor differences with the original clamped end condition.

Fig. 36 - The roller end condition

Fig. 37 - The deflection data for the modelled and the rigid part ("Roller" end condition)

Fig. 38 - The bending stress distribution in the modelled section ("Roller" end condition)

Fig. 39 - The bending stress distribution in the modelled section, the “Roller” end condition, (zoomed in section)
**CASE STUDY VI, BAR UNDER BENDING, RIGID SPRING VIRTUAL PART**

The problem in this case study is the continuation of the one discussed in the previous case. The cantilever beam of 150 mm is loaded with 1000 N load as shown in Fig.40. The plan is to discard the 50 mm section on the right hand side (ie not to mesh it) but to take into account the flexibility (stiffness) of this piece. Keep in mind that this section has a length \( d = 50 \) mm and the calculation must be based on this length. Referring to the coordinate system described in Fig. 40, we have

\[
k_{TZ} = \text{Translational stiffness} = \frac{3EI}{d^3} = \frac{3 \times (200 \times 10^9) \times (8.333 \times 10^{-10})}{0.05^3} = 4 \times 10^6 \text{ N/m}
\]

\[
k_{RX} = \text{Rotational Stiffness} = \frac{EI}{d} = \frac{(200 \times 10^9) \times (8.333 \times 10^{-10})}{0.05} = 3.333 \times 10^3 \text{ N.m/rad}
\]

In Fig.41, the Catia v5 dialogue box where the stiffnesses are inputted is shown. Note that in the present problem, other stiffnesses are ignored as there are no loading components in those direction. However, if one is interested, the calculations similar to the one presented above can be made and inputted in the dialogue box. All the needed formulas were presented in the first four case studies.
The theoretical deflections of the tip of the bar (i.e., at a distance of 150 mm, and 100 mm from the clamp) are calculated below.

For the true tip (150 mm away from the support),

\[
\delta_{150\,mm} = \frac{F(L + d)^3}{3EI} = \frac{1000 \times (0.1 + 0.05)^3}{3 \times (200 \times 10^9) \times (8.333 \times 10^{-10})} = 6.75 \times 10^{-3} \, m = 6.75 \, mm
\]

For the tip of the modelled part (100 mm away from the support),

\[
\delta_{100\,mm} = \frac{FL^3}{3EI} + \frac{(Fd)L^2}{2EI} = \frac{1000 \times (0.1)^2}{(200 \times 10^9) \times (8.333 \times 10^{-10})} \left[ \frac{0.1}{3} + \frac{0.05}{2} \right] = 3.5 \times 10^{-3} \, m = 3.5 \, mm
\]

The bending stress at the support (clamp) is calculated from,

\[
\sigma_{\text{wall}} = \frac{M_{\text{wall}}c}{I} = \frac{F(L + d)h}{I} = \frac{1000 \times (0.1 + 0.05) \times 0.005}{8.333 \times 10^{-10}} = 900 \times 10^6 \, Pa = 900 \, MPa
\]

The FEA generated deflection and the bending principle stresses are shown in figures 42 and 43 respectively and the results are in good agreement with the hand calculations above. Note that the end of the beam is assumed to be clamped and the stress distribution at the wall is presented.

Fig. 41 - The Catia v5 dialogue box for inputting the stiffness of the “Rigid Spring” virtual part

Fig. 42 - The deflection plot for the “Rigid Spring” model
CONCLUSION

In this paper, six case studies have been presented all of which are fairly well defined and understandable by any engineer (or engineering student). All that is required is a basic background in strength of materials. The theoretical calculations in these case studies were used as a basis for assessing the quality of the FEA results using “Rigid” elements. In all cases, there were good correlations between the two. More importantly, the case studies give the FEA users a much better understanding behind the rationale of the existence of such novelty elements.

Another point to raise is that the rigid and the “kind” of rigid assumptions require reasonable engineering judgment. Particularly, the “kind” of rigid element requires a good estimate of the stiffnesses involved. One cannot simply input ad/hoc numbers for such constants.

The virtual elements discussed in the present paper were employed for static problems, however they are equally applicable to dynamic situations. It would very beneficial to investigate these elements in the dynamic framework and report the results in an understandable way.

ACKNOWLEDGMENTS

The author gratefully acknowledges the support of the Faculty of Engineering at the University of Windsor under the internal grant 809153.

REFERENCES


