ISBN: 978-628-96613-3-0. ISSN: 2414-6390. Digital Object Identifier: https://dx.doi.org/10.18687/LACCEI2026.1.1.137

FEA Bolt Tightening Connections and Their Implementation in Catia v5™

Nader G. Zamani, Ph.D. P.Eng.
University of Windsor, Canada, zamani@uwindsor.ca

***Abstract*– The Catia v5 software has an embedded finite element solver known as the "Elfini" package specifically targeting the users in the early stages of the design process. Since bolted connections are very common in product development, such functionality is made available in three variations. These are known as Bolt Tightening, Virtual Bolt Tightening, and Virtual Spring Bolt Tightening Connections. This paper deals with some subtle issues regarding first two of these connections. The intention is shedding light on these connections particularly due to the scattered information or better put, the lack of it, in the public domain.**

***Keywords*—bolted connection, finite element analysis, virtual bolt, preload, stresses analysis**

I. Introduction

The access to highspeed computing at low cost has enabled industry to use CAD/CAE tools in the product development and the fabrication stages. It is highly unlikely that one comes across an engineering company which does not employ Computer Aided Design (CAD) software. Going back three decades, it becomes clear that the market was extremely crowded with CAD/CAM packages such as Catia [1], I-deas [2], Unigraphics [3], and Cadkey [4] to name a few. The situation has dramatically changed today. At this point, there are only three or four commercial outfits who are involved in this sector. One software who has survived the transformation is the Catia program (and its reincarnation) in the 3DEXPERIENCE platform by Dassault Systems. Catia v5 is till industry standard for solid and surface modeling and used by major corporations in the automotive, aerospace and robotics companies.

The primary clientele for the Catia program was the designers and that has not changed since its inception. However, roughly 20 years ago Dassault included a stripped-down version of a finite element program developed in France known as "Elfini" [5]. Although the stripped-down version greatly limits the abilities of the user to perform advanced nonlinear and Multiphysics simulation, it meets the need to employ FEA at the early stages of design. This drawback was rectified by the acquisition of the Abaqus program and bundling it with Catia v5 and marketing it with the label AFC (Abaqus for Catia) [6].

Since bolted connections are widely used in mechanically interacting components, some functionalities on this feature in the Catia "Generative Structural Analysis" workbench are provided. This workbench is where the finite element module lies within Catia v5. Unfortunately, the resources on how to perform the modelling with bolts are scarce and the software documentation is quite slim. It is not uncommon then to come across engineering designs that have employed these tools incorrectly. The main challenge of the software developers for bolted connections is to ensure that the pre-load in the bolt is properly modeled. When this was done in the early days, the concept of thermal expansion of the bolt leading to the correct strain and therefore correct bolt force was employed. It still is the primary algorithm behind modern commercial software.

II. Connections and Bolt Tightening Types

There are essentially two types of bolt modelling approaches in Catia. One possible approach is to physically model the bolt and include it as a meshed part in the calculation step. Although this approach is more accurate but if there is a moderately large number of bolts involved, it is not practical to model all. The second approach is to avoid the solid model creation of the bolt and instead introduce its functionality (clamping) using the so called "Virtual Bolt". Regardless of the choice, the first step is to introduce the "Analysis Connection" dictating which two components are carrying the bolt.

The depiction in Fig. 2 bundles the essential toolbars and icons needed to achieve these initial objectives. The "Analysis Support" toolbar and the particular icon used (circled in group (a) are presented. The grouping (b) indicates the "Face-Face Connection" toolbar where the actual bolt needs to be modeled. Finally, group (c) presented, deals with "Distance Connection" where the virtual bolt and virtual spring-bolt are displayed. In this paper the spring-bolt is not discussed.

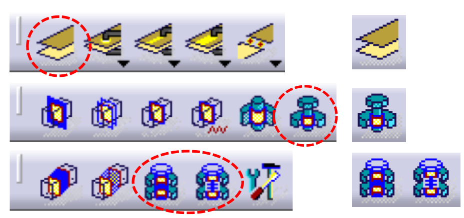

Fig. 1 (a) Analysis Supports (b) Face-Face Connection Property (c) Distance Connection Property

III. Bolt Tightening connections common questions

Due to the nature of the paper and the space limitations, the study is broad and not comprehensive however, the objective to discuss the pitfalls of bolt modeling are focused on. This is organized in the context of very simple examples avoiding the complications that can arise when the threads are included. This section deals with bolting two flat plates at a distance that are clamped at two edges as shown in Fig. 2. The bolt and nut are such that it is tightened to a bolt force of

500N. The dimensions of the plate, bolt, and nut are described in the figure. All components are made of steel operating in the linear elastic range, with Young's modulus of 200 GPa and Poisson ratio of 0.266. It is important to keep in mind that the FEA module within Catia v5 is not capable to handling material nonlinearities and the large strain situations.

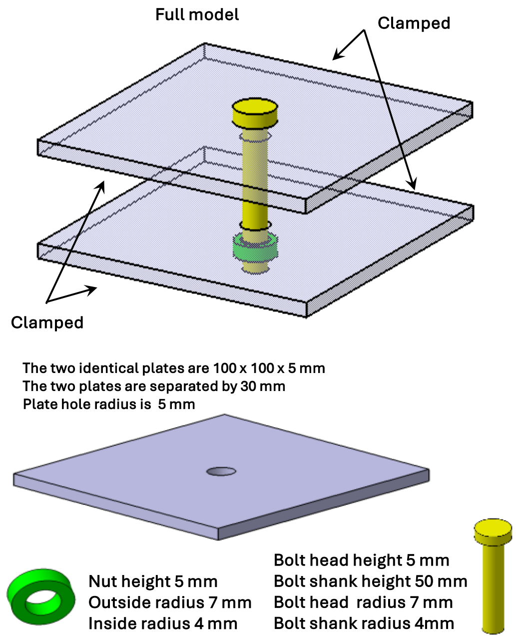

Fig. 2 Two steel plates clamped at the four edges are tied together as a pretensioned bolt/nut assembly

A common question that arises is, if planes of symmetry are used to reduce the FEA model, can they go through the bolts and if so, should the "Tightening force" be adjusted accordingly? This tightening force is specified in the dialogue box shown in Fig. 3. Catia help is very brief on the matter and no references in the public domain could be located. One should also be aware of the option "Orientation" in the dialogue box. In any problem, depending on what was done earlier, one of the choices works, and the other one will not. Basically the "icon" for bolt should correctly replicate the bolt/nut orientation.

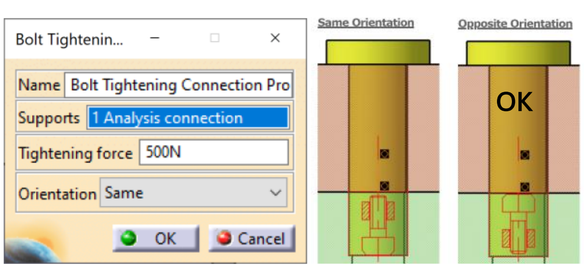

Fig. 3 Dialogue box and the proper orientation of the bolt

The above problem is solved three times, the full model, the half model, and quarter model. Every time a plane of symmetry is used, roller (surface slider) is employed regardless of whether, it is a bolt/nut that is cut or whether it is the plates. The two possible reduced models are shown in Fig. 4. At the same time, for the half and the quarter models, the "Bolt Force" reduced by a factor of 2. Therefore, they are set to 250N and 125N respectively.

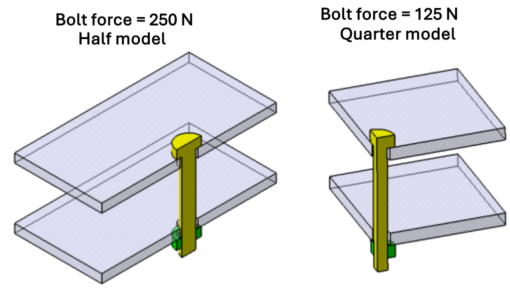

Fig. 4 Planes of symmetry used and bolt force is reduced

For discretization, a relatively coarse mesh is used but the elements are selected to be parabolic. The details of the values for the mesh are given in Fig. 5 for the components, namely, plate, bolt, and nut. Due to the nature of the paper and the space limitations in the presentation, these same values are used regardless of what model is being considered.

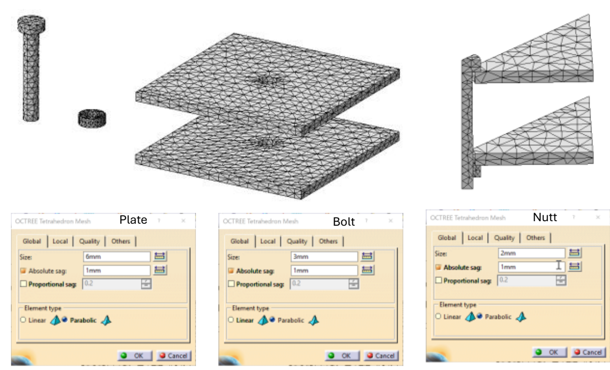

Fig. 5, Element specifications and sample mesh

It is important to emphasize the selection of the "first" and "second" components to create the "General Analysis Connection" is the first step of creating the "Bolt tightening" feature, Fig. 6.

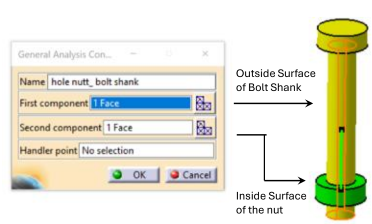

Fig.6 Order of component selection

One should note that if the order of the selection of components is reversed, the "Orientation" box of the dialogue box in Fig. 3 must also be consistent with the selection.
Now that the preliminary steps and introductory remarks are out of the way, the test cases considered are introduced.

IV. THE BENCHMARK 1, BOLT AND NUT MODELED

The results of full model in terms of the overall displacements under a bolt tightening force of 500N is shown in Fig. 7 (a). It is followed by figures (b) and (c), where the bolt forces have been accordingly reduced to 250N and 125N.
In figure 7(d), the bolt is also included and that is the reason behind the shift in the maximum contour legend. Obviously, the stretching of the bolt overwhelms the distance between the two plates.

It is worth noting that because friction between surfaces has been ignored in models (a) and (b), the rigid body motion had to be eliminated by introducing "Smooth Virtual" parts with appropriate restraints.

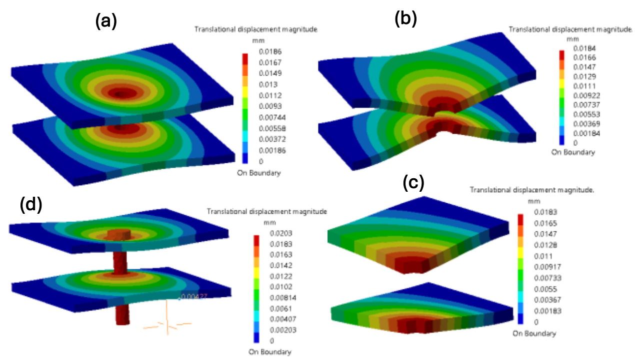

Fig.7 Displacement contour for the three models

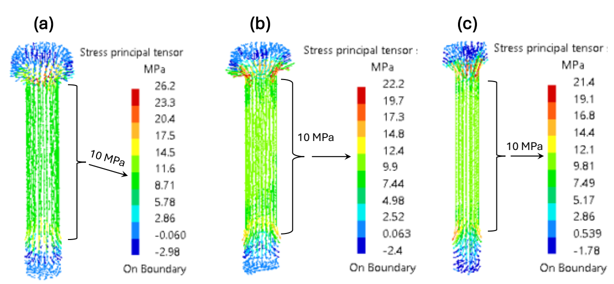

Fig.8 Axial Principal stress in the bolt for the three models

Note that the displacement for the three models are in close agreement based on the contour legends for all three models. The critical issue is that the bolt tightening force must be proportionally adjusted. In Fig. 8, the axial stress in the bolt based on the bolt force is plotted. This is done by plotting the appropriate principal stress value. The theoretical value can be calculated by dividing the bolt force by the bolt area (8mm diameter). This gives value 9.95 MPa, which is very close to the predicted Catia values. These predicted values are in the 10 MPa "ballpark".

The reader should be reminded that two other "Analysis Connections" were also created. The first one is used to define the contact condition between the bolt head and the top plate. The second connection is to enforce the contact between the top of the nut and the bottom plate. Because the diameter of the bolt was substantially smaller that the plates' holes, no contact condition was defined between the bolt shank and the plates.

V. THE BENCHMARK 2, VIRTUAL BOLT

In this section the strategy where the connection is defined without modelling the bolt (and nut, if present). This is clearly the desired approach if many bolts are involved. This approach in Catia v5 is referred to as the "Virtual Bolt Tightening" connection. Clearly this leads to some inaccuracies, for example the bolt head which interacts with the clamped pieces is not present. Issues like this give the user many options, each with their own advantages, and disadvantages. A quick glance at Fig. 9 indicates three choices (among others) that can be used for setting up the connection between the plates of benchmark 1 discussed earlier. The close-up view of the "Sewed" surface is displayed in Fig. 10.

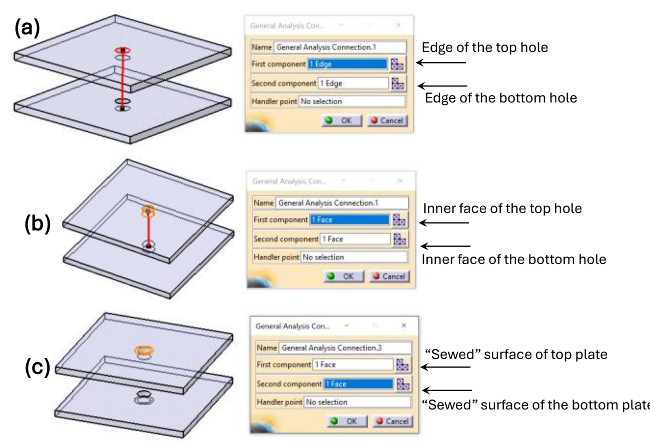

Fig.9 Three of the many possible choices for the analysis connection of the virtual bolt model

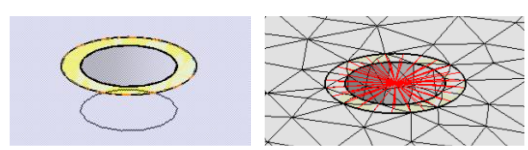

Fig.10 Close-up view of the "Sewed" surface

Note that introducing a "Sewed" surface will enable the engineer to emulate the size of the bolt head and the nut. Obviously, a corresponding Sewed" surface is at the bottom of the lower plate which accounts for the interaction with the nut (contact connection).

For the sake of completeness, the representative meshes for the full model and quarte model are shown in Fig.11. Please note that these are not the "Sewed" surfaces. The half model mesh is fairly similar to the ones depicted. The elements are parabolic and obviously a great deal of saving is involved as the nut and the bolt are not modeled.

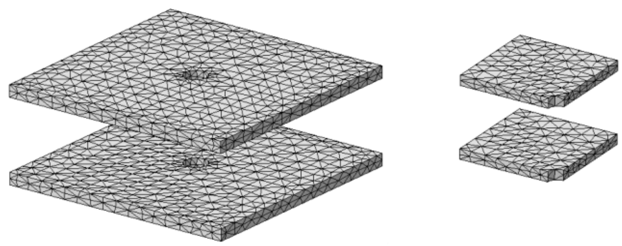

Fig.11 discretization of the full and quarter models

The dialogue box associated with the "Virtual Bolt Tightening" connection is shown in Fig. 12. In this dialogue box, a "Tightening Force" of 500N is specified. Therefore, it corresponds to the full model. When the half model and quarter model are run, this value must be changed to 250N and 125N respectively. Furthermore, because there is no bolt or nut, there is no rigid body motion, and no need to use "Virtual Part" for restraining. As for the "Analysis Connection", for the first and second components, the "ring" edges of the top and bottom holes are picked.

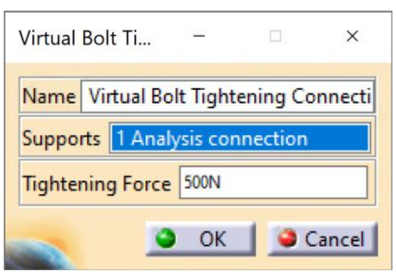

Fig.12 "Virtual Bolt Tightening" connection dialogue box

The displacement results of full, half, and the quarter models presented in Fig. 13. Notice that the maximum displacements are fairly close to each other, another indication that when the virtual bolt is used with symmetry considerations, the bolt tightening force must also be proportionally adjusted. In the present problem, the bolt forces are 500N, 250N, and 125N. These virtual bolts, all use the edges of the top and bottom holes to represent the "First" and "Second" components of the "Analysis Connection".

In this same figure, the plot at the bottom left corner corresponds to the situation where the two "Sewed" surfaces are employed in the "Analysis connection".

Comparing the maximum plate displacement associated with the three models clearly shows that the results are in good agreement. Furthermore, if one compares this with the scenario where the bolts were modeled (Fig. 7), there is only 5% difference between the two situations. However, there is significant computing overhead where the bolts are meshed and included in the analysis run.

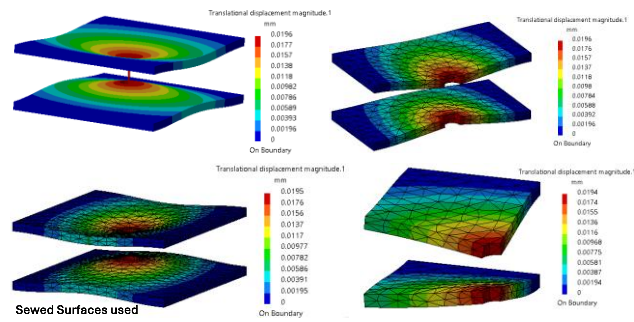

Fig.13 displacements result of the virtual bolt models

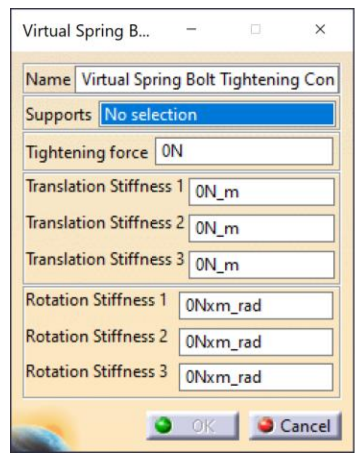

Fig.14 "Virtual Spring Bolt" definition dialogue box

It was indicated earlier that the "Virtual Spring Bolt" will not be discussed in the present paper. However, the reader can benefit from the shortened version of its role in a more realistic analysis. For this purpose, the text from the Catia documentation is presented below verbatim.

"The Virtual Spring Bolt Tightening Connection takes into account the elastic deformability of the surfaces and since bodies can be meshed independently, the Virtual Spring Bolt Tightening Connection is designed to handle incompatible meshes.". Needless to say, that in the dialogue box associated with this functionality, the stiffness of the spring needs to be specified as shown in Fig. 14.

VI. THE BENCHMARK 3, LOADING BOLTED CONNECTIONS

This last benchmark involves a situation where the parts that are clamped together are subjected to an external load. Two hollow shafts are clamped together with two identical bolts (and nuts) as shown in Fig. 15. The preload (bolt force) for each bolt is 500N. The bottom surface of the lower shaft is clamped, and the upper shaft is pulled by a force of 1000N.

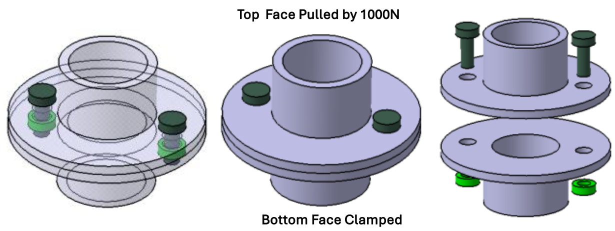

Fig. 15, Axially loaded clamped shaft assembly

Keeping in mind that one of the goals of the present paper is to confirm the bolt behavior when symmetry planes are used, the problem is solved with half and quarter models as described in Fig, 16. It is important to note that in the both models, the bolt can have a rigid body motion which needs to be taken care of. Once again, the "Smooth Virtual" is employed to enforce the necessary restraints. For the sake of completeness, the dimensions of the shaft assembly are provided in Fig.17 with the bolt and nut dimensions being identical to Benchmark 1.

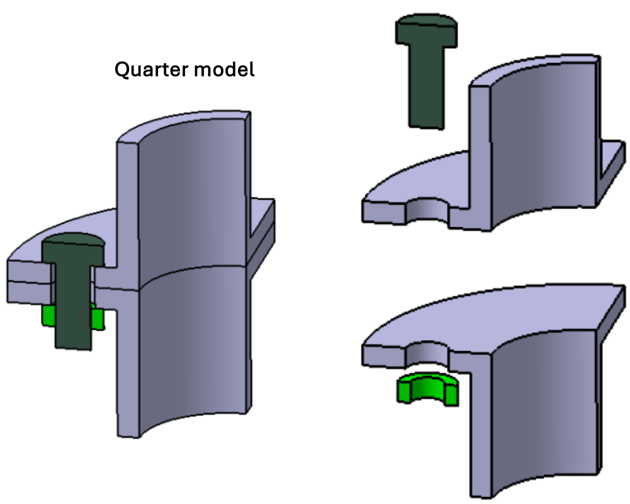

Fig. 16, Quarter model representation

In principle, the issue of rigid body motion and the usage of virtual parts to eliminate it can be completely avoided if a large enough friction coefficient is specified in all contact connections. This however is not done by the author. Such an approach adds another layer of complications which is deemed to be unnecessary for this presentation.

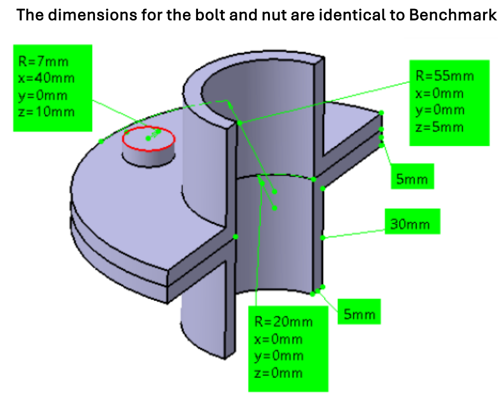

Fig. 17, the length dimensions of the bolted shaft assembly

The mesh associated with the quarter and half models with parabolic elements of size 3 mm are shown in Fig. 18. In this section it was decided to avoid working for the full model and only rely on the reduced models due to symmetrical considerations. As seen, the discretization is relatively coarse.

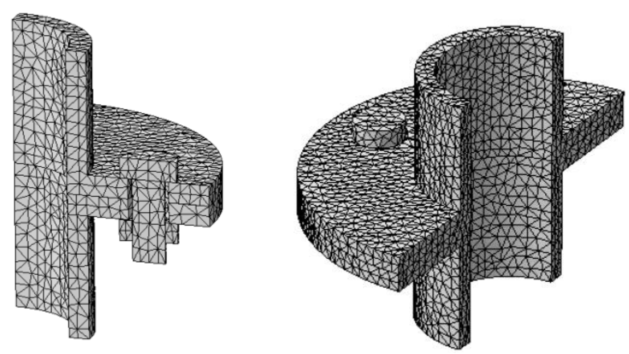

Fig. 18, The mesh discretization of the half and quarter models

The elements used are tetrahedral type which are fairly efficient in dealing with arbitrary geometric domains. Since the present geometry has a curvature, the so called "Sag" has an impact and therefore the dialogue box in presented in Fig. 19.

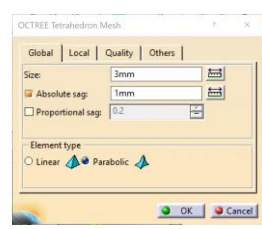

Fig. 19, Complete dialogue box for element specs

The displacement contours of the shaft assembly are shown in Fig. 20. The top two plots correspond to the half and quarter models, and contour legends indicate that the results are in good agreement. Therefore, this is another independent confirmation that the bolt force must be proportionally reduced. The bottom two contours in Fig. 20 depict the von Mises stress distribution in the bolt. They are also in reasonable agreement as expected. Note that in this problem, the bolts exhibit a behavior which was clearly missing in Benchmark 1. However, the clearance between the bolt shank and the holes is exactly 2mm which is significantly larger than the magnitude of the displacement. Therefore, no contact with the sidewalls of the holes is anticipated.

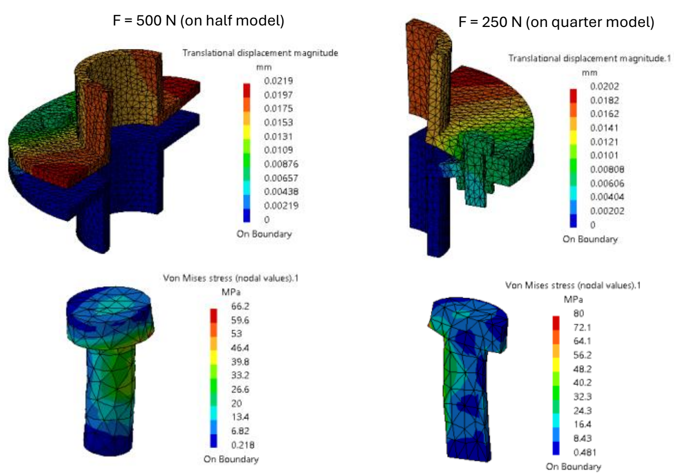

Fig. 20 Quarter and half models of the shaft assembly

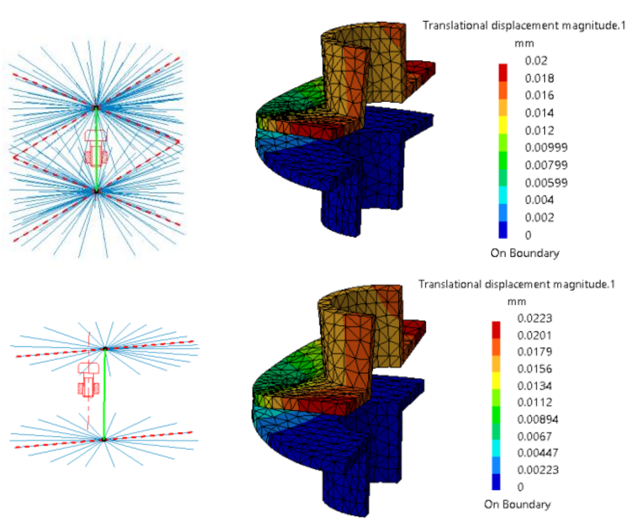

Fig. 21 Displacement contours of the half model using the virtual bolt tightening connection

Figure 21 shows the contours of the shaft assembly where the "Virtual Bolt Tightening" connections are used in the half model. The difference between the two contours is the nature of the "Analysis Connection". The top one involves the hole faces as the "First", and the "Second component". The bottom contour, however, is using the hole edges for this purpose. The results are in good agreement with each other and with the contours of Fig. 20.

As pointed out earlier, In the case where the bolt is modelled, it goes through two modes of deformation. The axial (tensile) modes, and the bending mode. Therefore, the stress based on the bolt force is exceeded. To have a better picture of what transpires, the axial principal stress is plotted in Fig. 22. In the shank section, this value ranges between 13 MPa and 25 MPa which are more than double the preload stress. In the introduction section, it was mentioned that Catia v5's finite element module is restrictive, although good enough for early structural integrity design.

Naturally, one cannot predict the outcome of elastic-plastic analysis and therefore, the failure is due to bolt fracture. However, infinite life fatigue failure of bolts can be handled (to some extent) with linear elastic finite element tool. Even the "Virtual Bolt Tightening" which avoids the meshing of the bolts is rather limited. Obviously if one in interested in the design of bolts as threaded connection, the requirements are completely different. That requires the discretization of the threads and their plastic deformation which are not possible with Catia v5.

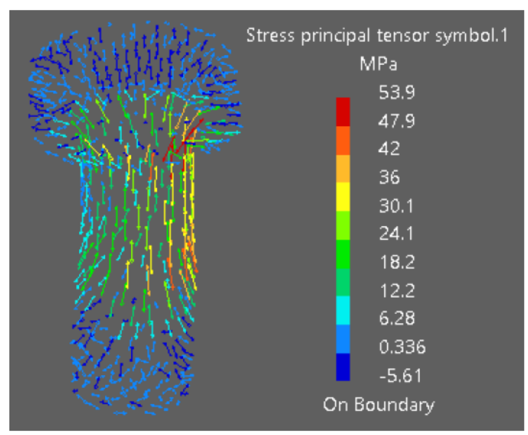

Fig. 22 Axial Principal stress in bolt with bending effect

VII. 3DEXPERIENCE CATIA AS AN ALTERNATIVE

In early 2000, Dassault Systems acquired HKS, the developer and proprietor of the Abaqus finite element software and bundled it with the Catia v5 package. The combo was marketed with the name Abaqus for Catia, but it had a short life span. Primarily because Dassault was simultaneously developing the 3DEXPERIENCE business platform [7], [8] through other acquisitions. The focus of DS was shifted to Product Life Cycle (PLM) and integrated business platforms.

The CAD workhorse integrated in 3DEXPERIENCE is Catia v6 (now referred to as the 3DEXPERIENCE Catia) and the setup is entirely cloud based. As a point of reference, Fig. 23 displays six items associated with bolts (or closely related items such as pin) within the R2025x of the 3DEXPERIENCE interface.

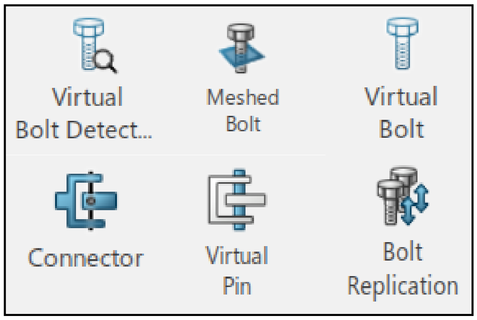

Fig. 23 Some "Bolt" related icons in the interface

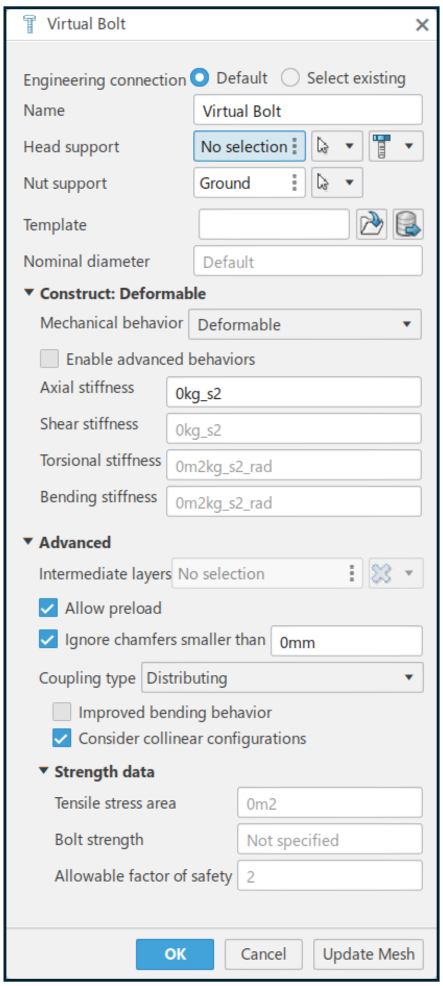

Fig. 24 Information for Virtual Bolt in 3DEXPERIENCE

The extent to which the enhancements are provided can be clearly seen by looking at Fig. 24, which describes the additional information that has to be inputted. It is quite an impressive list but arrives at a cost. It takes a great deal of effort by the user and involves a steep learning curve.

VIII. THE BENCHMARK 3, SHELL ELEMENTS AND BOLTS

Very frequently, solid elements are bypassed in favor of shell elements. This happens routinely in thin-walled pressure vessels, aerospace structures, and shear panels. Fasteners are commonly used for attaching components together. In such situations, the element of choice may be the "shell" elements meshing surfaces. For this reason, it is not possible to use actual physical bolts in the discretization. The way around that is to employ the "Virtual Bolt Tightening" functionality.

To explore the validity and of the strategy, the geometry of Benchmark 1 will be employed. The plate is meshed with parabolic elements as shown in Fig. 25 and the tightening connection is between the two ring edges on the holes. The displacement and von Mises stress contours for the full model are also depicted in the figure. Please note that "Virtual Bolt Tightening" functionally is associated with the connection and the bolt force is assigned to be 500N. Figure 26 displays the results for the "quarter" model and bolt force of 125N.

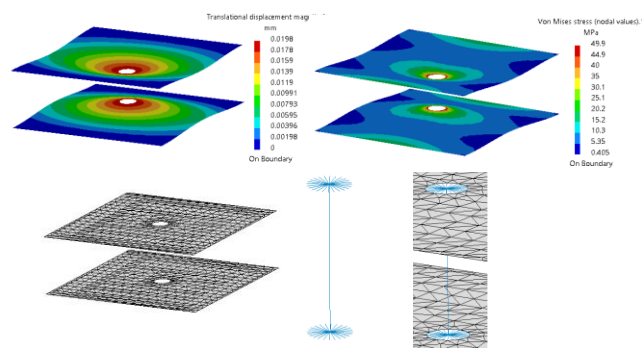

Fig. 25 The results for the "full" model

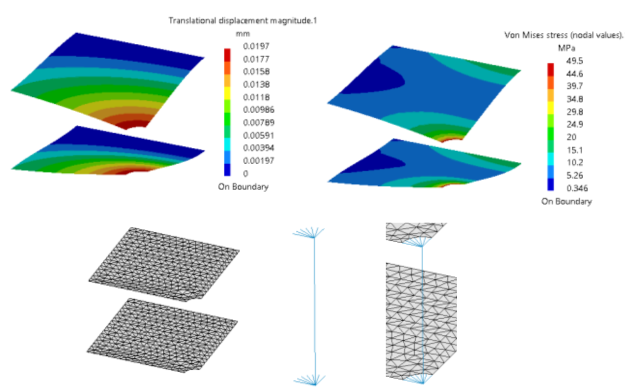

Fig. 26 The results for the "full" model

A quick comparison of the full, and the quarter models in Fig. 26 immediately reveals that the strategy adopted also applies to shell discretization. Furthermore, the results are in close agreement with the solid element discretization presented in Benchmark 1. If the bolt head size (and the nut size) may have a significant impact in the analysis, one can create local surfaces to emulate the contact interaction zones as described in Fig. 27 and employ them in the "Analysis Connection" dialogue box. The outcome of this extra effort may significantly improve the FEA results.

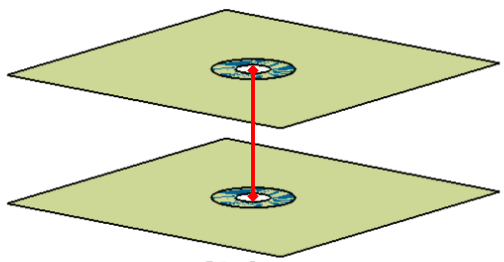

Fig. 27 Created local surfaces to account for the size of the bolt head and the nut

For the sake of completeness, the case of axially loaded shaft assembly is presented too. Here the shell elements are used to discretize the part and the "Virtual Bolt Tightening" connection is used to clamp the two pieces together without modelling the physical bolt. This is displayed in Fig. 28 together with the displacement contour. Note that the bottom edge is clamped and the top edge is pulled by a force of 1000N. Clearly, symmetrical boundary conditions are applied to the free edges.

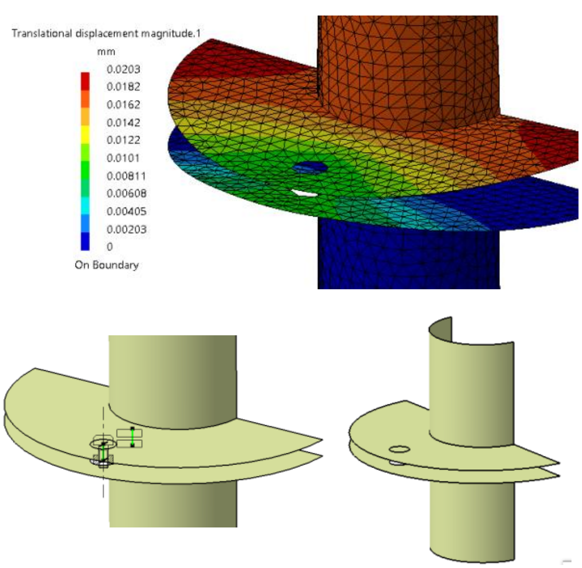

Fig. 28, An axially loaded shell assembly and Virtual Bolt

It is important to note that the two flanges of the top and bottom shafts are not to penetrate and therefore, a contact connection also needs to be specified. This is clearly visible next to the "fastener" icon in Fig. 28. The reader is reminded that since the symmetry planes are not cutting the "Virtual Bolt", the full bolt force of 500N is specified Unfortunately, no information about a bolt can be obtained, as there is no bolt.

VI. Final Remarks and Conclusions

The objective of this paper was to shed some light on certain aspects of the FEA solver embedded within the Catia v5 program. Due to the nature of the clientele, which is the designer's community, the capabilities are limited yet very relevant in the early design stages. The specific aspect of the finite element modelling in Catia which is investigated in the paper is the "Bolt Tightening" connections. There is very little information in the public domain which discusses the role of symmetry in bolt modelling.

Through several benchmarks considered, where the planes of symmetry are cut through the bolts. These benchmarks involve both the structure under external loads and free from any such loads. It becomes evident that the bolt forces must be proportionally reduced if they are cut. These statements apply to both situations where physical bolts are modelled, and where virtual bolts are employed.

Since not all structures are modelled with solid elements, for example the thin-walled structures, the adaptation of the strategy with shell elements is also investigated. The conclusion is the same, namely the bolt forces have to be proportionally reduced. The reincarnation of the Catia program in the 3DEXPERIENCE platform is also brought up. The finite element solver in this case is the well-known Abaqus program which was acquired and bundled with Catia in 2002. The bolt functionalities are considerably enhanced and can handle both material nonlinearities such as plasticity and creep. Furthermore, the strains are no longer restricted to infinitesimal values.

The author is very grateful to the constructive comments of the conference referees and has tried to incorporate the changes/modifications as requested. It is important to emphasize that the present paper has no claims on the convergence issues of the mesh. Furthermore, the accuracy of the predicted results is not the focus of this work. The main intention is to present a strategy of using bolt connections in the well established Catia v5 computer software. The reincarnation of Catia v5 in the 3DEXPERIENCE platform is a major achievement of Dassault Systems. It no longer relies on the extremely limited capabilities of the "Elfini" FEA software bundled with v5. One can now use the state-of-the-art FEA capabilities offered by the Abaqus program. It can also be interfaces with other CAE tools available throughout the platform. The only requirement being the proper "Role" as a user. As with other capabilities, one can name a few, such as, topology optimization [9], computational fluid dynamics

[10], and electromagnetism[11].Needless to say that there is a great demand or educational offerings by the company as there is a learning curve involved [12].

ACKNOWLEDGMENT

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

REFERENCES

[1] Catia v5, Dassault Systemes Corporation. Velizy, France, 2025.
[2] I-Deas, Structural Dynamics Research Corporation, Cincinnati and Milford Ohio 2001.
[3] Unigraphics, Electronic Data Systems, Detroit Michigan, 2000.
[4] CAD Key, Micro Control Systems Corporation, Parametric, Boston Massachusetts, 2010.
[5] N.G. Zamani, CATIA v5 FEA Tutorials, Release 21, SDC Publications, Mission Kansas, 2012.
[6] N.G. Zamani and Shuvra Das, Abaqus For CATIA, AFC v2, SDC Publications, Mission Kansas, 2008.
[7] 3DEXPERIENCE, Dassault Systems, Velizy-Villacoublay, France, 2025.
[8] Dassault Systems, White Paper, Links Management in 3DEXPERIENCE, Design and Engineering Applications, Velizy 2021.
[9] Dassault Systems, Enhancing Product Design with Topology Optimization, Online Store, Velizy 2025.
[10] J. Tu, G.H. Yeoh, C. Liu, and Y. Tao, Computational Fluid Dynamics, A Practical Approach, Elsevier 4th Edition, 2023.
[11] S. Humphries, Finite Element Method for Electromagnetics, CRC Press, 1st Edition,1997.
[12] N.G. Zamani, @naderzamani5752, YouTube, 2014-2026.