Analysis of bolted connection used in a reflectometry system for ITER

Diogo Miguel dos Santos Mendes Vilar Murteira
diogo.murteira@tecnico.ulisboa.pt

Instituto Superior Técnico, Universidade de Lisboa, Portugal
April, 2017

Abstract

The goal of this work was to propose and evaluate a methodology to perform thermal-structural analysis in bolted connections of a reflectometry system for ITER. The main concerns the analyses should cover are the connections’ alignment - given that they connect guides through which microwave radiation travels - and the comparison between stress distributions and stress limits with the ones defined by ITER. The analyses were performed in the Finite Element software Ansys Workbench Mechanical and the connection models were especially designed for this work using computer-aided-design (CAD). Both the analysis tools provided by the software and the modelling options used in the proposed methodology are described in detail. The methodology was firstly verified by applying it to solve problems for which analytical solutions are known, and then applied to perform the analysis of bolted connections under three distinct loading cases: connection subject only to bolt-tightening force, connection subject to bolt pre-tightening and imposed temperature distribution, and connection subject to bolt pre-tightening and “worst-case-scenario” static loading. The main conclusions were that the methodology proposed is able to provide converged results on bolt stress that can be compared to ITER defined stress limits and to provide some insight on waveguides’ alignment. It presents some problems regarding stress convergence in singularity points and contact regions.

1 Introduction

1.1 ITER Project description

The "International Thermonuclear Experimental Reactor" (ITER) is an international collaboration aiming to project the world’s largest Tokamak, a magnetic fusion device that has been designed to prove the feasibility of fusion as a large-scale and carbon-free source of energy [1]. ITER aims to be the first nuclear fusion device to achieve a positive net energy balance.

1.2 Plasma position reflectometry (PPR) system

ITER will include a vacuum vessel in which the nuclear fusion experiments take place. It will contain the plasma environment created by the fusion of deuterium and tritium (two hydrogen isotopes). The plasma will be magnetically kept from touching the vessel’s walls by a powerful magnet system. A number of diagnostics systems will provide information about plasma. The plasma position reflectometry system is a diagnostics tool that will serve to track the distance between the plasma and the vessel’s walls. Fig. 1.1 shows the configuration of the system.

The antenna is directly facing the plasma and has the function to detect information about the plasma’s position. The information is carried by microwaves from the antenna into the waveguiding system. To provide structural stability and to fix the system to the vacuum vessel, supporting structures are connected to it, and in turn connected to welding pads welded onto the vacuum vessel wall. For efficient manufacturing, the waveguiding system shall be comprised of several smaller waveguide segments, that are connected to each other and to the antenna by the bolted flange connections.

Fig. 1.1 - PPR system: antenna in pink, supporting structures in grey, waveguiding system in orange, waveguide covers in yellow, welding pads in blue, bolted flange connections in green.

1.2.1 Concerns over bolted connections

Waveguides’ alignment

Microwave radiation travels thought the antenna and the waveguiding system, which can be seen as rectangular hollow section beams (tapered in the case of the antenna), with curvature defined by previous studies on the subject. Consequently, both the zones of connection between antenna and waveguides and between two waveguide segments are the most dangerous regarding the possible disruption of the microwaves path. If the interior edges of two
connected segments are not aligned to the micrometer order of magnitude, microwaves could either escape or not be transmitted correctly, compromising the diagnostics’ operations.

**Stress Limits**

Design criteria are defined by ITER [2] to prevent structural damage in the connections’ fasteners. Compliance with Stress Limits is therefore desirable in all loading cases to which the structure may be subjected (i.e., initial tightening, thermal loading, etc.). In addition, fastened members should not undergo excessive plastic deformation. The defined limits are:

\[
\bar{\sigma}_m \leq \left\{ \begin{array}{l}
0.9\sigma_Y(T_m) \\
0.67\sigma_u(T_m)
\end{array} \right. \\
\sigma_m + \sigma_b \leq \left\{ \begin{array}{l}
1.2\sigma_Y(T_m) \\
0.9\sigma_u(T_m)
\end{array} \right.
\]

where \( \bar{\sigma}_m \) and \( \sigma_m + \sigma_b \) are the mean and maximum Von-Mises (VM) stress in the bolt, neglecting stress concentrations. \( \sigma_Y \) and \( \sigma_u \) are the material yield and ultimate strengths. \( T_m \) is the temperature.

1.3 Motivation

This work resulted from the need to find a suitable methodology for the Finite Element Analysis (FEA) static structural and thermal-structural analyses of bolted connections present in the PPR system. The proposed methodology should be able to produce accurate results regarding stress and displacement distributions on the areas of interest within those connections. It should also be elaborated according to ITER imposed requirements, such as making use of CATIA® [3] and Ansys® [4] CAD and FEA software products.

1.4 Aerospace Engineering framework

The motivation to include this study in a Master’s Dissertation in Aerospace Engineering resides in the fact that all the project’s tools, analysis methods and analytical calculations are the same as the ones used for projecting joints of the same nature in aerospace applications. Examples of such are the analysis and comparison of steel and carbon fiber bolted joints applied in rocket motor casings, carried out by Rajamani and Gupta [5] through the FEA of developed 3D CAD models, where it is concluded that Carbon fiber composite T-700 proves efficient in such joints both from the structural and thermal standpoints. Ouskouei, Keikhosravv and Souts [6] have also performed FE analysis on aluminum bolted joints loaded in tension and present in aircrafts and concluded that tightening fasteners to hold the joint with high preloads can reduce damaging effects of the bearing stress at the fastener hole under high longitudinal tensile loads. Iremann, Nyman and Hellbom [7] present a variety of design and analysis methods for bolted joints in composite aircraft structures and conclude that while fast methods such as design diagrams and analytically based computer programs will continue to prove more useful in the design of the large amount of joints present in an aircraft, FEA of 3D models presents useful for critical primary joints. Mc Carthy and Mc Carthy [8] wrote a book chapter on design and failure analysis of composite bolted joints in aircrafts in which FEA of these joints is presented and where it is concluded, among other things, that bolt-hole clearance does not have a significant effect on the ultimate failure load of joints.

2 Theoretical Background

2.1 Solid Mechanics – Linear Elasticity

For linear elastic isotropic materials, stress and strain tensors are symmetric, and can be expressed, in Voigt notation, as [9]:

\[
\sigma = [\sigma_x \sigma_y \sigma_z \tau_{xy} \tau_{yz} \tau_{xz}]^T
\]

\[
\varepsilon = [\varepsilon_x \varepsilon_y \varepsilon_z \gamma_{xy} \gamma_{yz} \gamma_{xz}]^T
\]

If small deformations are assumed, strain relates to displacement by:

\[
e = Du
\]

where \( u \) is a displacement vector and \( D \) is a matrix differentiation operator defined as:

\[
D = \begin{bmatrix}
\frac{\partial}{\partial x} & 0 & 0 \\
0 & \frac{\partial}{\partial y} & 0 \\
0 & 0 & \frac{\partial}{\partial z}
\end{bmatrix}
\]

Hooke’s law is a constitutive equation that relates a material’s stress state to strain, as follows:

\[
\sigma = E\varepsilon
\]

where \( E \) is defined as:

\[
E = \begin{bmatrix}
\lambda + 2\mu & \lambda & \lambda \\
\lambda & \lambda + 2\mu & \lambda \\
\lambda & \lambda & \lambda + 2\mu
\end{bmatrix}
\]

in which:

\[
\lambda = \frac{\nu E}{(1 + \nu)(1 - 2\nu)} \quad \text{and} \quad \mu = \frac{E}{2(1 + \nu)}
\]

\( E \) is the material’s Young’s Modulus and \( \nu \) is the Poisson’s coefficient.
If a body is in equilibrium, neglecting inertial forces, than the following relation is verified:

\[ D^T \sigma + X = 0 \]  \hspace{1cm} (2.8)

### 2.2 Thermoelasticity

Note that \( \varepsilon \) can be viewed as [10]:

\[ \varepsilon = \varepsilon_s - \varepsilon_T \]  \hspace{1cm} (2.9)

where \( \varepsilon_s \) contains strains caused directly by deformations and \( \varepsilon_T \) contains strains caused by other factors, such as thermal expansion.

For a three-dimensional body with known thermal field \( T = T(x,y,z) \), if \( \Delta T_s = T - T_0 \), where \( T_0 \) is defined as a reference state of uniform temperature distribution which causes zero stress or strain in the body, then the thermal strains are expressed as follows:

\[
\varepsilon_T = \left[ \begin{array}{c}
\varepsilon_{xT} \\
\varepsilon_{yT} \\
\varepsilon_{zT} \\
\varepsilon_{xyT} \\
\varepsilon_{xzT} \\
\varepsilon_{yzT}
\end{array} \right] = [a \Delta T_x \ a \Delta T_y \ a \Delta T_z \ 0 \ 0 \ 0]^T
\]

where \( a = \alpha(T) \) is the material's coefficient of thermal expansion.

Assuming that both \( a \) and \( E \) are not affected by temperature in this case, (2.5) can take the form:

\[ \sigma = E(\varepsilon_s - \varepsilon_T) \]  \hspace{1cm} (2.11)

#### 2.2.1 Coupled Thermal load by weak coupling

If a weak coupling is assumed (temperature distribution is assumed not to be influenced by deformation), than equilibrium equation (neglecting inertial forces) can be written as:

\[ D^T \varepsilon (Du - \varepsilon_T) + X = 0 \]  \hspace{1cm} (2.12)

If \( \varepsilon_T \) is known, than it can be viewed as known boundary condition of the displacement type [9].

#### 2.3 Bolted Connections' member stiffness

For symmetric connections in which the fastened members are made of the same material, stiffness of those members can be calculated by [11]:

\[
k_m = \frac{P}{\delta} = \frac{t \tan \varphi m Ed}{2 \ln \left( \frac{t \tan \varphi 2d + d_w - d}{d_w + d} \right) (d_w + d)} \]  \hspace{1cm} (2.13)

where \( d, D \) and \( \varphi \) are defined as shown in Fig. 2.1, and \( D \) is the fastener's washer face diameter, \( d_w \).

### 3 Methodology

#### 3.1 Verification of the procedure used in the software

Ansys Workbench 17.1 (AWB) was used for analysis, and particularly its “Mechanical” feature (MECH). As such, verification of the FEA results in the software was necessary. For this purpose, some simple problems for which analytical solutions are known were selected and modeled in order to compare FE results to analytical solutions.

##### 3.1.1 Thermal-structural verification

The problem of a rectangular plate with dimensions of 2x1x0.2 m³, subject to a known \( \Delta T \) under two different sets of boundary conditions (BCs) was chosen:

Set 1: The plate has frictionless supports on plane: \( x = 0, y = 0, \) and \( z = 0; \Delta T = 78 K \).

Set 2: Frictionless support on plane \( x = 0, y = 0 \) and \( y = 1 m, \) and plane \( x = 0 \) and \( x = 0.2 m; \Delta T = 78 K \).

The CAD model for geometry was created and imported into MECH to generate its FE model and solve the FE problem.

#### 3.1.2 Static structural verification with bolted joint

The purely structural problem chosen for verification purposes was that of a simple concentrically bolted joint of two plates, only subject to pretension in the bolt. A 3D CAD model of two concentric plates fastened by a bolt and nut was created and is presented in Fig 3.1.
3.2 PPR system bolted connections analysis method
The method used for the analysis of the connectors in the reflectometry system follows the following steps:
1. Examination of the existing 3D CAD models.
2. Selection of parts of interest for analysis.
3. Alteration of individual models, when necessary.
4. Importation of altered geometries into AWB.
5. Modification or defeaturing of geometries, in AWB's CAD software SpaceClaim Design Modeler (SCDM).
6. Creation of FE model for imported geometries in MECH.
7. Setup of Boundary Conditions for analysis.
8. Solving FE problem with Mechanical APDL solver.
9. Analysis of results of interest.

3.2.1 Examination and modification of CAD models
The assembled model for the PPR system is presented in Fig. 1.1. It contained incomplete models of the flanges connections studied in this work and shown in Fig. 3.2 note that both models didn’t include fasteners and b) didn’t include bolt holes.

![Fig. 3.2 – Initial CAD models for flanges connections: a) connection 1 (C1); b) connection 2 (C2)](image)

Consequently, new CAD models were created in SCDM (presented later in Fig. 3.5-a and 3.5-b. The fastener elements chosen for the flanged connections are simplified models of: M8 ISO 4014 Bolts, M8 ISO 4032 Nuts, and M8 ISO 7089 Washers.

Simplifications assumed when modelling (to facilitate meshing) were: bolts’ heads and nuts hexagonal geometries simplified to circular with diameters equal to their washer faces.; gaps between bolts and flanges holes and between bolts and washers neglected; bolt’s body extends exactly from head to the plane of lower face of nut; thread’s geometries are not modeled in CAD, but instead in MECH’s contact definition (see section 3.2.3 – Contact Definition).

3.2.2 Defeaturing
Some of the selected individual CAD models contained features, such as fillets or extra-edges, which are known to originate computationally expensive meshing processes. For that reason, most of the individual parts composing the connections studied where subject to defeaturing in order to eliminate these types of elements.

3.2.3 Generation of FE model in MECH
Material Properties Definition
The material model was created in AWB in an Engineering Data instance. At this stage, projected material for all parts in C1 and C2 is Stainless Steel Type 316 LN - IG (SS 316), with properties defined in [12]. Since its material model is not available in the program’s library, IPFN’s team created one in XML file format which was imported.

Coordinate Systems
Coordinate systems objects (CSOs) can be defined in MECH by selecting geometric features present in mechanical models, such as edges, faces, bodies, or mesh nodes and elements.

Symmetry region
This feature allows the user to define a plane of symmetry by selecting a planar geometrical feature, i.e. face. The model, including boundary conditions, is then solved as symmetric.

Contact definition
Since the analyzed connections are assemblies, there was the necessity to define the way in which they interact with one another when in contact. MECH is set by default to automatically find contact regions between bodies. This is done by defining a tolerance length under which faces from different bodies are in contact. Contact regions can also be defined manually, by selecting contacting faces. In this study, contact regions were mostly defined either to “frictional” or “frictionless”, as follows:

“Frictional” – Used in all contact regions between horizontal faces (i.e. bolt head to flange, top flange to bottom flange, etc.). The option was chosen because, in a real situation, the bodies contacting each other will experience friction when subject to the types of loading analyzed. The static friction coefficient was set to $\mu = 0.15$, which fits in the range found in [13] for “Stainless Steel to Stainless Steel” static friction.
“Frictionless” – Used in all contact regions between bolts shanks and flanges and between bolts and washers, like in Fig. 3.3. Since gaps between those parts are neglected, and in a real situation, there would be no physical contact between these regions, “frictionless” contact was set on them.

Bolt-Thread Contact Geometry Correction
“Bolt-Thread Contact Geometry Correction” was chosen as the thread model for all fasteners present in this work. To use this feature, first the bolt’s cylindrical faces were split by an edge contained in the plane defined by the nut’s upper face. In MECH, “Bolt Thread” was defined for the contact between those faces, as illustrated in Fig. 3.4. Then the mean diameter, pitch and thread angle were defined.

Note that thread modelling via the “Bolt Thread contact Geometry Correction” is expected to produce stress results that are qualitatively, but not always quantitatively, correct [14]. While a CAD model of a detailed thread would be more realistic and required for a detailed analysis of stress in the thread, it would also be much more computationally expensive, both in meshing and in solving the models [14], [15].

Meshing
Quadratic, or 20-noded hexahedral elements (HEXA20) were chosen to mesh most of the tridimensional geometries analyzed in this work, since the problems analyzed involve non-linear contact. To achieve good quality, the “Multizone method” was chosen, when possible, as the semi-auto meshing method.

3.3 Application of Analysis Method
The goal is to study results for VM stress and displacement distributions in 3 cases (Fig 3.5-a to 3.5-f), in order to evaluate the method’s ability to produce meaningful results and to perceive its limitations.
A comparison of results with the main concerns expressed in section 1.2.1 will be presented. Application cases include:

**Case 1**: Study conducted to C1 in which 4 different values for pretension were applied to the bolts: preload $P$: (1) $P = 4000 N$; (2) $P = 7000 N$; (3) $P = 8000 N$; (4) $P = 10000 N$. To prevent rigid body motion the supports seen in Fig 3.5-d were also applied.

**Case 2**: Study conducted on C2 with preload $P = 7000 N$, an imposed temperature distribution shown in Fig 3.6 and supports shown in Fig 3.5-e.

The reason to include temperature as a load applied to C2 is the fact that this type of connection is the one used to connect antenna and waveguides, and is therefore the flanged connection that is closest to the plasma. As such it will experience the largest temperature gradients.

**Case 3**: Study conducted to C1, when connecting two portions of covers and pads. These are connected to portions of covers and pads, as shown in Fig 3.5-c. Pretension $P = 7000 N$ is applied to bolts. "Worst-case-scenario" forced displacement loads, taken from [16] and supports are shown in Fig 3.5-f.

### 4 Results and Discussion

#### 4.1 Plate with imposed temperature

Considering the problem described in section 3.1.1. To analytically calculate strains and stresses, eqs. 2.4 to 2.11 are used:

**BC set 1**: Since the only load applied to the plate is $\Delta T = 78 K$ and BCs keep it free to deform in all three directions, stresses and elastic strains ($\varepsilon_{x/y/z} E$) are 0. Thermal strains are:

$$\varepsilon_{xt} = \varepsilon_{yt} = \varepsilon_{zt} = \alpha(T - T_0)$$

(4.1)

$$= 1.2 \times 10^{-5} \times (78) = 9.36 \times 10^{-4}$$

**BC set 2**: In this case the plate is only free to deform in $x$ direction. In this case, thermal strains will be the same as above. Furthermore, since the plate is free to deform in the $x$ direction, $\varepsilon_x = 0$ and $\varepsilon_y = 0$. Note also that there are no deformations allowed in $y$ and $z$ directions and thus $\varepsilon_{yE} = \varepsilon_{zE} = 0$. $x$ and $y$ direction normal stresses are:

$$\sigma_y = \sigma_x = \frac{E}{1 - \nu} \left( \varepsilon_{yE} + \nu \varepsilon_{zE} \right) - (1 + \nu) \alpha (T - T_0)$$

(4.2)

$$= -267.429 MPa$$

#### 4.1.1 FEA of "plate with imposed temperature" in AWB

**Mesh for “plate with imposed temperature” FEA**

The mesh for analysis is presented in Fig 4.1.

**Results for “plate with imposed temperature” FEA**

For both sets of BCs, the thermal strain presents a constant value of $9.36 \times 10^{-4}$ throughout the plate, equal to the analytically calculated one.

For the BC set, as seen in Fig 4.1 normal stresses in the $y$ and $z$ directions present the value of $-267.43 MPa$, which is less than 0.1% apart from the one calculated analytically.

#### 4.2 Fastened concentric circular plates

Consider the problem described in section 3.1.2. To calculate the members’ deflection, $\delta_m$, due to pre-load $P$, their stiffness, $k_m$, is calculated by (2.13) and noting that all the members are made of the same material. Cone angle $\varphi$ is obtained from (2.14):

$$\tan \varphi = 0.570479 \Leftrightarrow \varphi = 29.7^\circ$$

(4.3)

$$k_m = \frac{0.570479 \pi \cdot 2 \times 10^5 \frac{N}{mm^2} \cdot 12}{2 \ln \left( \frac{(0.570479 \cdot 27.76 + 18 - 12)(18 + 12)}{(0.570479 \cdot 27.76 + 18 + 12)(18 - 12)} \right)}$$

(4.4)

$$= 2.47788 \times 10^6 \frac{N}{mm}$$

#### 4.2.1 FEA of “fastened concentric plates” in AWB

In order to solve the problem in AWB MECH, the CAD model created in SCDM was imported into MECH.
Meshing for “fastened concentric plates” FEA
Mesh for analysis is presented in Fig 4.2.

Results for “fastened concentric plates” FEA
Since member stiffness is not available as result, to compare analytical and FEA results, equation (2.13) is used. The axial force applied is \( P = 2000\,\text{N} \), and the deformation distribution is known, so it is possible to calculate \( k_m \).

Due to the 3D nature of the geometry, the value for \( \delta_x \) was found by computing its average over the surfaces of the washers which are in contact with the bolt and nut washer faces. The calculation of \( k_m \) follows:

\[
k_m = \frac{2000}{\delta_x\text{(avg)}_1 + \delta_x\text{(avg)}_2} \times 10^6 \, \text{N/\text{mm}}
\]

Comparing with the analytical stiffness, the error of the FEA is 5.8453%.

4.3 Method application studies
4.3.1 Results of Method application studies
Displacement Distribution
Displacement results are shown in Figs 4.3-4.5, corresponding (by the same order) to the three cases mentioned in section 3.3.

In case 1, maximum occurs in the bolts, due to the preload, which forces the bolt’s body to elongate. In case 2, an ellipsoid-like pattern “emanates” from the vertex where fixed support is applied, and the maximum occurs in the top of bolts. This is consistent with thermo-elasticity theory and the temperature distribution seen in Fig 3.6 (which shows higher values in the top of flange and in the bolt's head). In case 3 displacement patterns follow the forced displacement load, and maximum occurs in the pads on which it is applied. Maximum values for displacement were subject to convergence study presented in Fig. 4.6-4.8.

Note that meshes shown in Fig. 3.5-d to 3.5-f correspond to meshes no 4, 3 and 4 of cases 1, 2 and 3, respectively. Convergence is observed in
case 2. Case 3, although not strictly convergent, presents a maximum variation of 0.008% in $u_{max}$ between successive refinements.

**Waveguides Alignment**

Alignment of the waveguide’s segments is guaranteed by the flanges, so it must be ensured that loads do not produce significant relative motion between the top and bottom flange interior edge, at the interface between the two.

To track the distance between these edges under the considered loading cases, displacement was computed in points along each edge, like shown in Fig. 4.9.

![Fig. 4.9 – Edges for tracking alignment](image)

In this way, maximum non-alignment in x and y direction ($\text{dist}(\text{x})_{\text{max}}$) and $\text{dist}(\text{y})_{\text{max}}$, and maximum z direction separation ($\text{sep}(\text{z})_{\text{max}}$) were retrieved for all loading conditions. Table 4.1 shows, for case 1 - mesh 4, alignment results for all values of preload.

<table>
<thead>
<tr>
<th>$P$ (N)</th>
<th>4000</th>
<th>7000</th>
<th>8000</th>
<th>10000</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\text{dist}(\text{x})_{\text{max}} \times 10^{-3}$</td>
<td>1.47</td>
<td>5.27</td>
<td>6.55</td>
<td>9.51</td>
</tr>
<tr>
<td>$\text{dist}(\text{y})_{\text{max}} \times 10^{-2}$</td>
<td>3.1013</td>
<td>5.295</td>
<td>6.189</td>
<td>7.708</td>
</tr>
<tr>
<td>$\text{sep}(\text{z})_{\text{max}}$</td>
<td>0.666</td>
<td>1.143</td>
<td>1.304</td>
<td>1.629</td>
</tr>
</tbody>
</table>

Table 4.1 – Alignment study for case 1 with all values for $P$

Table 4.2 shows the comparison between values for the 3 cases, with preload $P = 7000\text{N}$.

<table>
<thead>
<tr>
<th>$\text{dist}(\text{z})_{\text{max}}$</th>
<th>Case 1</th>
<th>Case 2</th>
<th>Case 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\text{dist}(\text{x})_{\text{max}}$</td>
<td>$4.87 \times 10^{-4}$</td>
<td>$1.701 \times 10^{-4}$</td>
<td>6.652</td>
</tr>
<tr>
<td>$\text{dist}(\text{y})_{\text{max}}$</td>
<td>$4.543 \times 10^{-2}$</td>
<td>$1.612 \times 10^{-2}$</td>
<td>5.508</td>
</tr>
<tr>
<td>$\text{sep}(\text{z})_{\text{max}}$</td>
<td>1.129</td>
<td>9.597x10^{-1}</td>
<td>3.259</td>
</tr>
</tbody>
</table>

Table 4.2 – Alignment study comparison of cases 1,2,3

Note, in case 1, that the increase of preload augments the magnitude of non-alignment. Furthermore, note that, for cases 1 and 2, although geometries are slightly different, non-alignment values present similar. In case 3, non-alignment presents the largest, which is expected due to “worst-case-scenario” loading. Fig.4.10 shows the separation zone in flanges for case 1.

![Fig.4.10 – Separation zone for case 1/Results magnification factor 6.5x10^2](image)

**Stress Distribution**

The Von-Mises (VM) stress distribution for the 3 cases considered (all with preload $P = 7000\text{N}$) is shown in Fig. 4.11-4.13.

![Fig. 4.11 – VM stress distribution for case 1](image)

Von-Mises stress distributions present similarities in all three cases. Main differences are in maximum values. Note also, in case 3, that the areas of contact between waveguides and flanges’ rectangular holes show larger stresses than the other two cases, which should be due to the transmission of load between waveguides and flange.

Table 4.3 presents maximum stress values for the 3 cases, for the meshes shown in Fig. 3.5-d to 3.5-f.
Convergence was studied for results presented in Table 4.3. It was not verified. In fact, the analysis of stress results shows that all the parts in the connection present areas in which results don’t converge:

“Frictional” contact interfaces – The reason may be mesh related, more specifically the difference between meshes of bodies in contact, i.e. surface nodes in one body are not coincident with nodes in the other body. Examples of these are the region of contact between bolt head and flange, particularly in the elements that are in direct contact.

Corner below bolt’s head – As expected, bolt elements in the immediate vicinity on this corner show a singularity-like stress evolution in consecutive mesh refinements. Sharp reentrant corners are a known cause of stress singularities [17]–[19].

Thread zone – For the reasons explained in section 3.2.3, “Bolt thread contact” is the most probable cause for non-convergence.

Corners of bodies in contact – Another known cause for stress singularities [17]–[19].

**Bolt Stress vs ITER defined limits**

To compare bolt stress with ITER limits defined in section 1.2.1, the values for mean and maximum Von-Mises stress were tracked in cross-sections along the bolts’ bodies. 20 cross-sections were analyzed in C1 for cases 1 and 3, and 15 cross-sections were studied in C2 for case 2. These cross-sections are the intersection of “surfaces” shown in Fig. 4.14 with the bolts’ bodies.

Note in Fig 4.14, that no cross-sections in the bolt’s head and thread were considered. The latter because it is a non-convergent zone, and the first because it is a low stress zone. For cases 1 and 2, temperature of all bodies is 20°C, which is the reference temperature for thermal expansion of SS316. As such, ITER limits for these cases are $\sigma_{\text{m}} < 0.9\sigma_t$ (SS316@20°C) = 198 MPa and $\sigma_{\text{m}} + \sigma_{\text{p}} < 1.2\sigma_t$ (SS316@20°C) = 264 MPa. Converged results for these quantities are $\sigma_{\text{m}} = 139.41$ MPa, $\sigma_{\text{m}} + \sigma_{\text{p}} = 155.91$ MPa for case 1 and $\sigma_{\text{m}} = 139.52$ MPa, $\sigma_{\text{m}} + \sigma_{\text{p}} = 160.55$ MPa for case 3. For case 2, due to the imposed temperature distribution, cross-sections present different mean temperatures between them, and consequently different stress limits apply to each. Table 4.4 shows the converged measured values for stress in each of the 15 cross-sections studied, and the corresponding stress limits.

<table>
<thead>
<tr>
<th>Sec. no</th>
<th>Measured stress (MPa)</th>
<th>Limit (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$\sigma_{\text{m}}$</td>
<td>$\sigma_{\text{m}} + \sigma_{\text{p}}$</td>
</tr>
<tr>
<td>1</td>
<td>121.94</td>
<td>121.46</td>
</tr>
<tr>
<td>2</td>
<td>121.94</td>
<td>124.1</td>
</tr>
<tr>
<td>3</td>
<td>121.95</td>
<td>124.27</td>
</tr>
<tr>
<td>4</td>
<td>121.95</td>
<td>124.41</td>
</tr>
<tr>
<td>5</td>
<td>121.94</td>
<td>124.48</td>
</tr>
<tr>
<td>6</td>
<td>121.73</td>
<td>124.4</td>
</tr>
<tr>
<td>7</td>
<td>120.76</td>
<td>125.03</td>
</tr>
<tr>
<td>8</td>
<td>118.21</td>
<td>121.47</td>
</tr>
<tr>
<td>9</td>
<td>121.94</td>
<td>123.79</td>
</tr>
<tr>
<td>10</td>
<td>121.96</td>
<td>123.64</td>
</tr>
<tr>
<td>11</td>
<td>122</td>
<td>123.45</td>
</tr>
<tr>
<td>12</td>
<td>122.08</td>
<td>123.1</td>
</tr>
<tr>
<td>13</td>
<td>121.9</td>
<td>123.39</td>
</tr>
<tr>
<td>14</td>
<td>119.64</td>
<td>122.17</td>
</tr>
<tr>
<td>15</td>
<td>110.45</td>
<td>127.82</td>
</tr>
</tbody>
</table>

Table 4.4 – Comparison between stress and ITER limits

Note that stress values from sections 1 and 9-14 are greater than the limit. This indicates that, although the bolt experiences stress relaxation from the increase in temperature, the limit values decrease at a “faster” rate, due to the decrease in yielding stress of the material.

**5 Conclusions and Future work**

A method for the analysis of bolted connections present in ITER’s PPR system was presented, with bidirectional CAD modelling and FEA as its main tools.

The numerical methodology proposed for the FE software was verified relatively to known analytical solutions. It was verified that FEA results are very close to analytical solutions. Three different structural analyses were performed to bolted flange connections of the PPR system: (1) - C1 with Pretension, (2) - C2 with Pretension and imported Body Temperature and (3) - C1 with pretension in “worst-case-scenario” loading. These analysis were focused in the obtaining of displacement and VM stress distributions. Displacement analysis provides some insight about potential problems in waveguide alignment. Stress analysis shows that there are some zones (some contacting surfaces, corners of bodies in contact and corners of bolts in which stress results require a treatment of singularities resulting from defecturing and contact definition. More detailed geometric models would be necessary to account for stress concentrations and accurate stress in threads. Contact definition originated problems that require further study. These seem restricted to the closest points to the contacting surface, and the transmission
of load between the different parts in the assembly seems sufficiently accurate.

Results that are considered potentially relevant for IFPEN’s Engineering and Integration Systems team include:

- Waveguide’s alignment and the stress limits imposed by ITER can be studied using the proposed analysis methodology.

- “Wort-case-scenario” loading that is calculated as an equivalent static load does not have a significant effect in stress distributions of fasteners, but produces potential issues regarding alignment of waveguides.

To continue the work presented in this text, one should consider:

- Studying the influence of boundary conditions on the analysis where only connectors are present (i.e. without waveguides).

- Studying the influence of mesh defeaturing and contact definition options on convergence of stress results.

- Performing comparisons between the traditional design analysis (by hand) and FEA.

6 Acknowledgements

I would like to express my thanks to:

My supervisors, Professor Miguel de Matos Neves and Dr. Hugo Policarp, for their guidance, availability to help, for their revision of this thesis and the support they gave me throughout its completion.

Instituto de Plasmas e Fusão Nuclear as an institution, for the trust laid on me and for choosing me to work in this project.

The work leading to this publication has received financial support from Fundação para a Ciência e Tecnologia (FCT) through project UID/FIS/50010/2013 through grant BL58/2016. Author further acknowledge the data provided by Fusion for Energy under F4E-FPA-375-04.

7 References