Design and Analysis of CFETR CS Coil and Pretension Structure

Zitong Liang\textsuperscript{1, a, †}, and Langzhou Jin\textsuperscript{2, b, †}

\textsuperscript{1}College of Mechanical and Electrical Engineering, Central South University, Changsha, Hunan Province, 410083, China.
\textsuperscript{2}Walter Scott, Jr. College of Engineering, Colorado State University, Fort Collins, Colorado State, 80521, United States.
\textsuperscript{a}ltz6017@163.com, \textsuperscript{b}davidjin@colostate.edu

\textsuperscript{†}These authors contributed equally.

Abstract. The coil and the pretension rod play important roles in the central solenoidal coil system of Tokamak. Therefore, the analysis of the coil and the pretension rod in the normal operating environment is essential for the facility’s construction. In this paper, the 3D model of the two parts mentioned above is built based on SolidWorks. Subsequently, the simulations are carried out to revert the real operation environments following the fields, loads, and boundary conditions. Then, we compare the official analysis results of the corresponding parts in the central solenoidal system of the China Fusion Engineering Test Reactor (CFETR) based on the results from COMSOL. Compared with the official analysis, the error is about 1.3\% for the maximum magnetic field and about 0.53\% error for the outer part, respectively. As for the pretension rod, simulations are carried out based on the finite element analysis software ABAQUS. The Von mises and the maximum principal stress are 140.1 MPa and 168.4 MPa, which is much lower than the material’s yield stress (205 MPa). The safety factor is 1.78, larger than the critical failure value 1. According to our results, the rationality of the CFETR central solenoidal coil system design is re-examined and verified.

1. Introduction
The Tokamak is a toroidal system with an axially symmetrical shape, in which the high temperature plasma is contained by the magnetic field of an inner current circulating [1]. Besides, a strong longitudinal magnetic field parallel to the current is used to suppress the main magnetohydrodynamic instabilities [1]. The central solenoidal (CS) coil system is an important component in the Tokamak to provide the poloidal magnetic field. The system has several main parts to support the normal operation, including the coil, the pretension mechanism and other important parts. In the CS coil system, the coil will generate the poloidal magnetic field to support the plasma in the reaction chamber. On the other hand, the pretension mechanism will ensure stability with the accompanied specific loads and boundary conditions during the operation.

Due to the important roles of the CS coil system, the analysis is necessary to investigate various loads during the working process to ensure normal and safe operation. The specific analysis aims at the chosen parts in the system, e.g., the coil and the pretension rod will generate the key ideas about the precautions. The integral cognition of the properties and the conditions for the entire system could be generated after
researching every component in the system. Thus, the generated and refined cognition will be meaningful insight to later researchers who study designs of the CS coil system.

The coil and the pretension rod in the China Fusion Engineering Test Reactor (CFETR) CS coil system are chosen as the analysis objects to fulfil this goal. In this analysis, the basic 3D models of the two parts are built based on SolidWorks. The different kinds of fields and loads are generated to create the simulations for a real operation environment to check whether the design is reasonable. Finally, with the comparison between the simulations and the official analysis results of CFETR, it is confident to claim the CFETR CS coil system design is valid. In this paper, section 2 makes a brief introduction to the research method, including the software platforms, guiding research ideology. In section 3, results, and discussions of the two objects in the paper are displayed in detailed descriptions, respectively. Specifically, we demonstrate the pre-setting, concept concretization, data generating methods, and visible results. Section 4 gives a brief summary based on the results eventually.

2. Methods
In this paper, several software platforms, including Solidworks2017, COMSOL5.6, and ABAQUS, are utilized to investigate different analysis parts, respectively. Specifically, SolidWorks, a solid modelling computer-aided design (CAD) and computer-aided engineering (CAE) computer program, is utilized to construct the 3D model of CS coil, the preload system, and the specific pretension rod [2]. COMSOL Multiphysics is a cross-platform finite element analysis, solver, and multi-physics simulation software used to create the magnetic field simulation around the coil and outer coil conductor [3]. For the pretension rod, ABAQUS, a Finite Element Analysis (FEA) software for finite element analysis and computer-aided engineering, is utilized to generate the potential loads and boundary conditions under the real operation situation [4].

2.1. Formatting the title Solidworks2017 setting/parameters
Solidworks2017 creates a coil part drawing with the parameters listed in Table 1 (collected from Ref. [5]). Based on the coil size, three-dimensional parts drawings of the tension rod, pressure plates, buffer zone, and load beam are established, respectively. Finally, the parts are assembled to obtain the general assembly drawing of the CFETR CS coil and its pre-tensioning mechanism, prepared for subsequent structural, magnetic field, and force analysis.

| Parameter                  | Nb3Sn Coil         | NbTi Coil         |
|----------------------------|--------------------|-------------------|
| Conductor Size/mm          | $49^2 \times 32.6$ | $49^2 \times 32.6$ | $51.9^2 \times 35.3$ |
| Conductor Insulation/mm    | 2.6                | 2.6               | 2.6               |
| Ground Insulation/mm       | 4.9                | 4.9               | 4.9               |
| Radial Turns               | 4                  | 4                 | 10                |
| Axial Turns                | 28                 | 29                | 22                |
| Total Turns                | 112                | 116               | 230               |
| Inner Radius/mm            | 750                | 963.8             | 1217.6            |
| Outer Radius/mm            | 953.8              | 1167.6            | 1760              |
| Height/mm                  | 1442.2             | 1493.8            | 1196.4            |
| Conductor Length/m         | 600                | 777               | 2060              |

2.2. Magnetic field analysis by COMSOL5.6
The magnetic field is analysed by COMSOL5.6. Firstly, we construct a geometric model of the coil where the two-dimensional cross-sectional view of the coil is illustrated in Figure 1. Then, a rotational symmetry operation is performed to obtain a three-dimensional CS coil diagram, as shown in Figure 2.
Besides, it is feasible to build three-dimensional models of single-turn coils with different diameters layer by layer. Nevertheless, the COMSOL 5.6 system will freeze if too many three-dimensional models are added, i.e., the former way is chosen in our case. Besides, the air domain is added with software default attributes. Materials and the magnetic field properties are entered following Table 1. Afterward, we construct the finite element mesh according to the parameters in Table 2.

![Figure 1. Coil two-position cross section and air domain diagram](image1.png)

![Figure 2. 3D model of CS coil in COMSOL 5.6](image2.png)

### Table 2. Finite element parameter

| Parameter type                  | Numerical value |
|---------------------------------|-----------------|
| Maximum cell size               | 400 mm          |
| Minimum cell size               | 0.08 mm         |
| Maximum unit growth rate        | 1.1             |
| Curvature factor                | 0.2             |
| Narrow area resolution          | 1               |
2.3. The strength analysis for the pretension rod based on ABAQUS FEA Method

2.3.1. Background for ABAQUS.
ABAQUS is an FEA program with various functions designed specifically for high level simulation analysis, including structure, heat transfer, etc. ABAQUS is also designed for both nonlinear and linear stress analysis of different kinds of structures with versatility [4].

For the pretension rod strength analysis, the finite element analysis (FEA) is used based on the ABAQUS platform. With the guideline of FEA, the research project will be divided into numerous target elements to make a simulation. The user defines the properties of the element (e.g., shape, quantity, etc.). The different kinds of load and boundary conditions will be added with demands to the research project. With the reaction from loads and boundary conditions, the simulation based on the meshed project will be created as the simulation.

2.3.2. The settings of the simulation
For the material of the pretension rod, the stainless steel Grade 316LN (UNSS31653) is chosen because of the versatility. The existed design has a similar dimension to the CFETR. Other corresponding properties (e.g., the density, Young’s modulus, etc.) are based on the properties of the chosen material. The adopted properties are all based on the ABAQUS unit system to fit the calculation and analysis based on the ABAQUS. The default length unit is in millimeters. The detailed data settings are shown in Table 3.

Table 3. The basic properties for the pretension rod

| Properties                  | Values      | Unit          |
|-----------------------------|-------------|---------------|
| Density                     | 7.99×10^9   | Tonne/mm^3    |
| Young’s Modulus             | 200000      | MPa           |
| Poisson’s Ratio             | 0.28        | N/A           |
| Thermal Expansion Coefficient | 17.2×10^-16 | 1/ °C         |

2.4 The reaction settings on the pretension rod simulation

2.4.1 Loads and boundary conditions
Three kinds of load are considered in the stimulation: gravity, temperature, and pretension. For the pretension load, each pretension rod will be added to 3.125 MN for the load. The temperature load is assumed to decrease from room temperature (293 K) to 4 K. The gravity load is generated autonomously with the initial density input. For the boundary condition, since any kinds of displacement or rotation are not allowed for the ends of the pretension rod, the boundary condition “encastre” is chosen. The limitation of the present studies naturally includes the restricted licence of the ABAQUS, i.e., the electromagnetic loads will not be presented here.

2.4.2 Strength analysis category choice
The simulation could display different kinds of results with the demands of properties. For the pretension rod, the Von mises and the maximum principal stress will be chosen as the reference to be compared with the original stress properties of the material. Then, the deformation on the three axes (U1, U2, U3, corresponding to x, y, z axis) will be considered at the same time to check if the unacceptable deformation occurs.
3. Results and Discussion

3.1 3D Modelling results of CS coil and pretension system

3.1.1 3D model of preload system and draw assembly diagram

![3D models of (a) inner and (b) outer coils from SolidWorks 2017](image)

Figure 3. 3D models of (a) inner and (b) outer coils from SolidWorks 2017

In order to achieve the design goals of the CS coil and reduce the manufacturing cost simultaneously, the coil is designed as a hybrid magnet coil with an Nb3Sn conductor in the inner high-field area and a NbTi conductor in the outer low-field area. Therefore, the coil winding design adopts the embedded package structure, which is divided into Nb3Sn coil module and NbTi coil module, and then packaged together. The two-dimensional cross-section design of the coil winding is shown in Figure 3. Generally, a total of 5 pie coils are constructed. Considering that the longest conductor length needs to be less than 1000m, the Nb3Sn coil module is wound and designed into two pie coils (namely the Nb3Sn inner coil and the Nb3Sn outer coil). Besides, NbTi, the coil module, is designed into three pie coils: NbTi upper coil, NbTi middle coil, and NbTi lower coil.

3.1.2 3D model of preload system and draw assembly diagram.

According to the preload structure provided by reference [5], we build the 3D model of preload system and assemble them with the coil models. Figure 4 illustrates the composition of the pretension system, including Upper and Lower Load Beam, Buffer Zone, Tension Rod, Pressure Plate. And it can be divided into 12 symmetrical structures.

![The assembly diagram of CS coil and its preload system: (a) three-dimensional assembly diagram and (b) assembly section structure drawing.](image)

Figure 4. The assembly diagram of CS coil and its preload system: (a) three-dimensional assembly diagram and (b) assembly section structure drawing.
3.2 Magnetic field simulation result

3.2.1. Finite element network & Magnetic field distribution around CS coil
The finite element network and the corresponding distribution of the magnetic field are shown in Fig. 5 and 6. The magnetic field distribution on the coil section under 49kA working current. The maximum magnetic field of the coil is 12.25T, which is near the inner center conductor of the inner coil. The maximum magnetic field on the outer coil conductor is 5.958T, located on the conductors at the upper and lower ends of the innermost layer.

![Figure 5. Finite element network](image)

![Figure 6. Magnetic field distribution](image)

3.2.2. Comparison with the results of reference [5]

Table 4. Magnetic field comparison with ANSYS [5] and COMSOL results

|                         | COMSOL | ANSYS | Program |
|-------------------------|--------|-------|---------|
| Maximum magnetic field of CS coil/T | 12.25  | 12.05 | 12.09   |
| Maximum magnetic field on outer coil conductor/T | 5.958  | 6.0   | 5.99    |

It can be seen from Table 4, the results of modeling and simulation using COMSOL are in agreement with the results of analyzing and programming using ANSYS. In detail, simulation results of ANSYS
show that the maximum magnetic field of the coil is 12.05 with 1.66% error while it is 12.09T with 1.32% error in our results. The maximum magnetic field of the outer coil conductor obtained by ANSYS simulation is 6.0T with 0.70% error, while it is 5.99T with 0.53% error in our results.

3.3 The results of pretension rod strength analysis

Fig. 7-11 shows the simulation results for the stress (Von Mises and Maximum Principal) and deformation (U1, U2, U3), respectively. From Fig. 8 and 9, the max stress of the Von mises stress simulation and the maximum principal stress simulation is 140.1 MPa and 168.4 MPa correspondingly.
8

Figure 11. The deformation of pretension rods in the U3 direction (collected from ABAQUS)

| Von mises stress | Maximum principal stress | Material yield |
|------------------|--------------------------|----------------|
| Maximum stress occurred (MPa) | 140.1 | 168.4 | 205 |

Table 5. The simulation stress results compare with the UNSS31653

For better verification of the stability of the pretension rods, we compare the results of the proposed method with those of the UNSS31653 yield stress. According to Table 5, both the simulation results of stress are lower than the UNSS31653 yield stress, i.e., reveal the safety and reliability of the current pretension rod design. On the other hand, based on the fact for the ductile material, the safety factor calculation is applied for the pretension rod with the maximum distortion energy theory:

\[
SF = \frac{\sigma_{\text{yield}}}{\sigma_{\text{Von mises}}} = 1.78 > 1
\]  \hspace{1cm} (1)

Since the safety factor is higher than the critical failure value 1, the design of the pretension rod is again proved to be safe and reliable enough.

4. Conclusion

In summary, the simulations are carried out for the coil and the pretension rod with the determinate field, loads, and boundary conditions to show the situation under a real operating environment. We identified the magnetic field around the pre-setting coil 3D model and working current, which align with prior literature. Meanwhile, the feasibility of the coil design has been further verified. We investigated three kinds of loads for the pretension rod and one boundary condition on the pretension rod. With the generated simulations, we found that the simulation result of the maximum magnetic field of the coil and the outer coil conductor has a high similarity with the official analysis results. For the pretension rod, the Von mises stress, and the maximum principal stress are all in the acceptable range of the material yield stress. The safety factor is also larger than the critical failure value. The specific analysis on the coil and the pretension rod of CFETR provides the basic cognition about the parts design and testing. In the future, the CS coil is looked forward to be an advanced design based on improving the innovation of the wiring, material, etc. With the help of the optimized FEA method, the more accurate and reasonable simulation closer to the real operation environment is possible to create, i.e., paves a path to improve the CS coil system.

References

[1] Artsimovich, L. A. (1972), Tokamak devices, Nuclear Fusion 12.2: 215.
[2] Schmitz, Barbara M. (1999), SolidWorks Animator, Computer-Aided Engineering.
[3] Multiphysics, C. O. M. S. O. L. (1998), Introduction to COMSOL Multiphysics®, COMSOL Multiphysics, Burlington, MA, accessed Feb 9: 2018.
[4] Abaqus, G. (2011), Abaqus 6.11, Dassault Systemes Simulia Corporation, Providence, RI, USA.
[5] Liu, Xiaogang, et al. (2016), Electromagnetic optimization and preliminary mechanical analysis of the CFETR CS model coil.” IEEE Transactions on Plasma Science 44.9:1559-1563.
[6] Wei, Cai, et al. (2019), Magnetic field analysis of solenoid driven by alternating current. Journal of Physics: Conference Series. 1237:3.

[7] Börgesson, L. (1996), Abaqus, Developments in geotechnical engineering, 79: 565-570.

[8] JI, Feng, et al. (2015), Magnetic field and electromagnetic load calculation for CFETR central solenoid model coil, Journal of Hefei University of Technology (Natural Science) 2015: 06 (Chinese).