Intro

- This presentation highlights the work done in developing a simple demo model

- It involves a thermal/cooling analysis

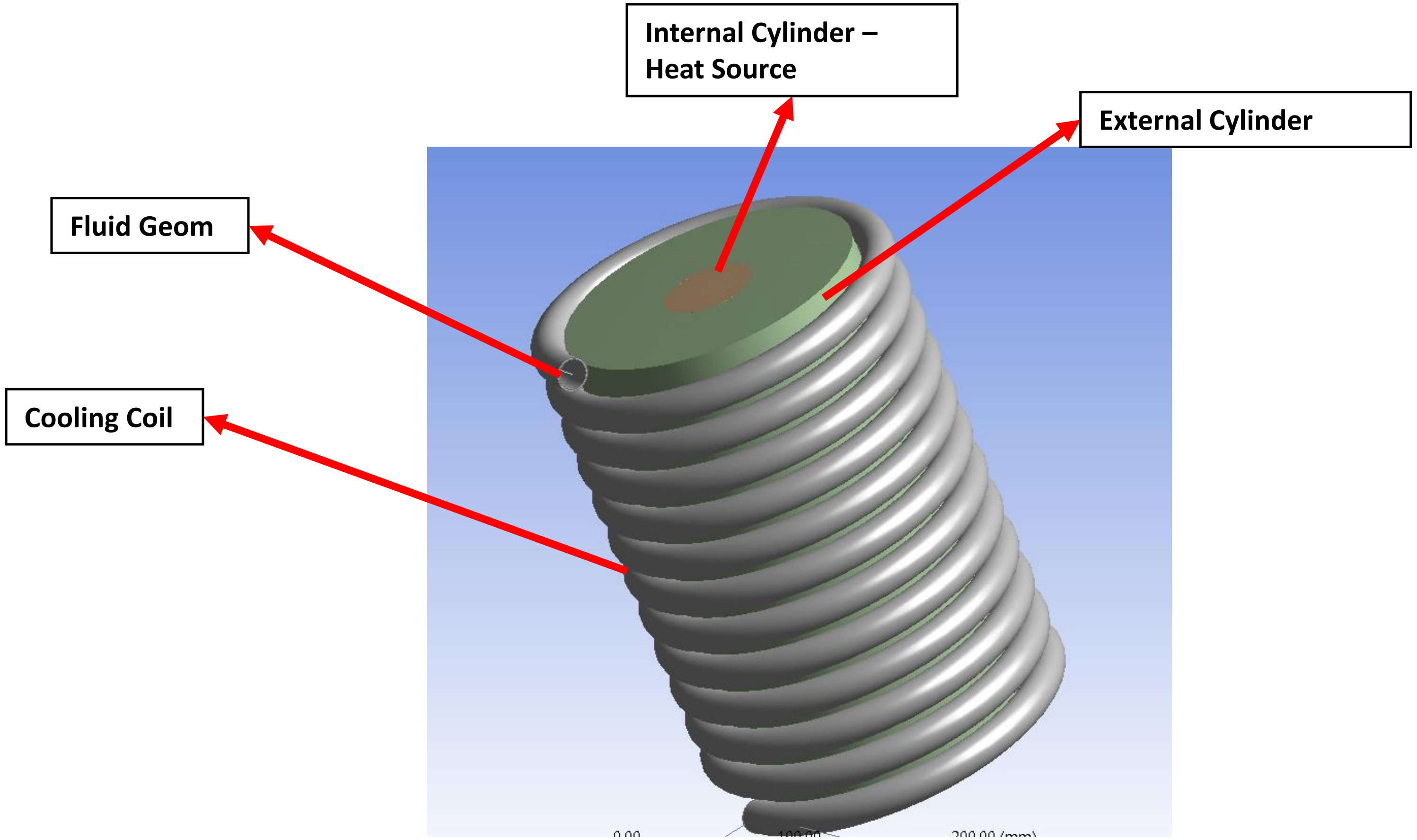
- The goal of this model (project) is to come up with a simple model in order to outline the capabilities of the Software

Initial CAD Geometry

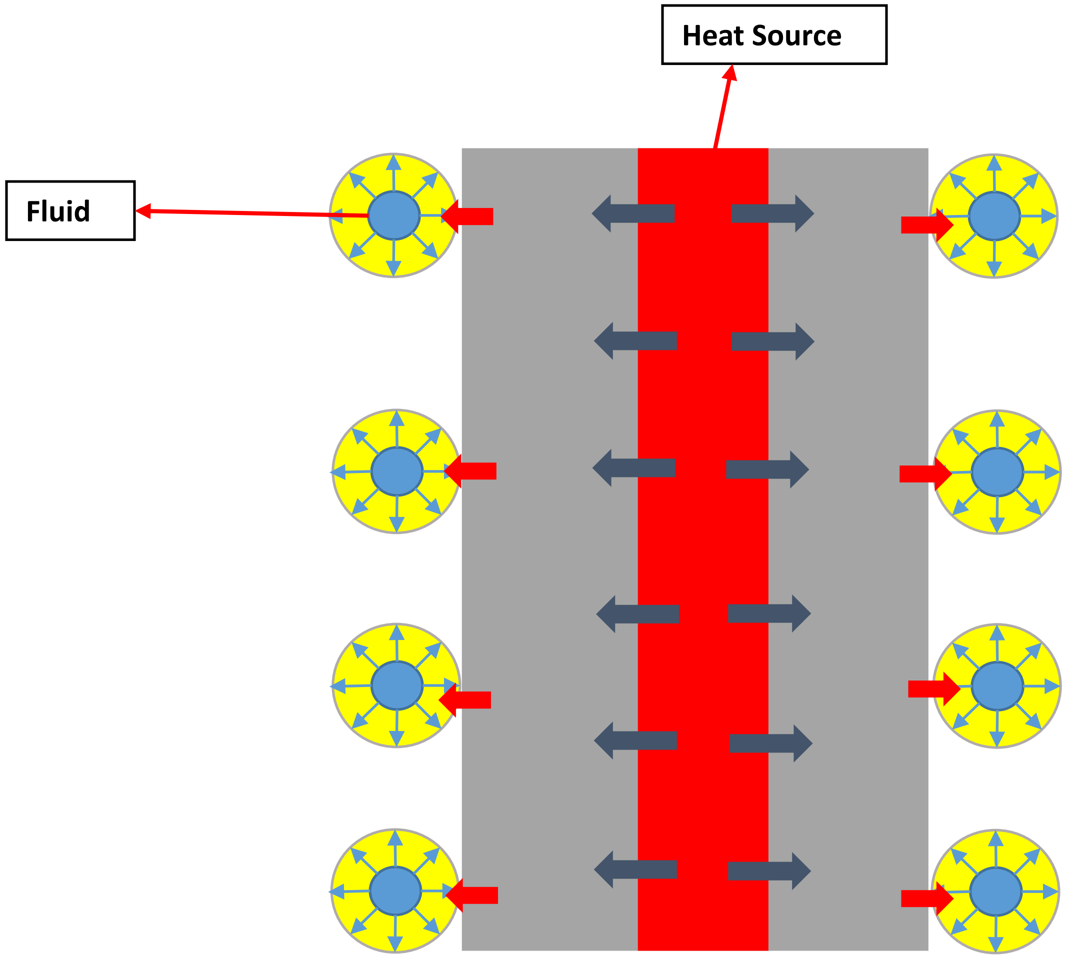
- The problem involves a cooling system. The customer's idea is to have a cooling coil attached around a solid cylinder – Heat source
- The tentative shape of the cooling coil is shown
- A heating source has a diameter of 54 mm while the external diameter of the cylinder is 178 mm
- The cooling coil has an internal diameter of 20 mm and a thickness of 1 mm

Simplified Model

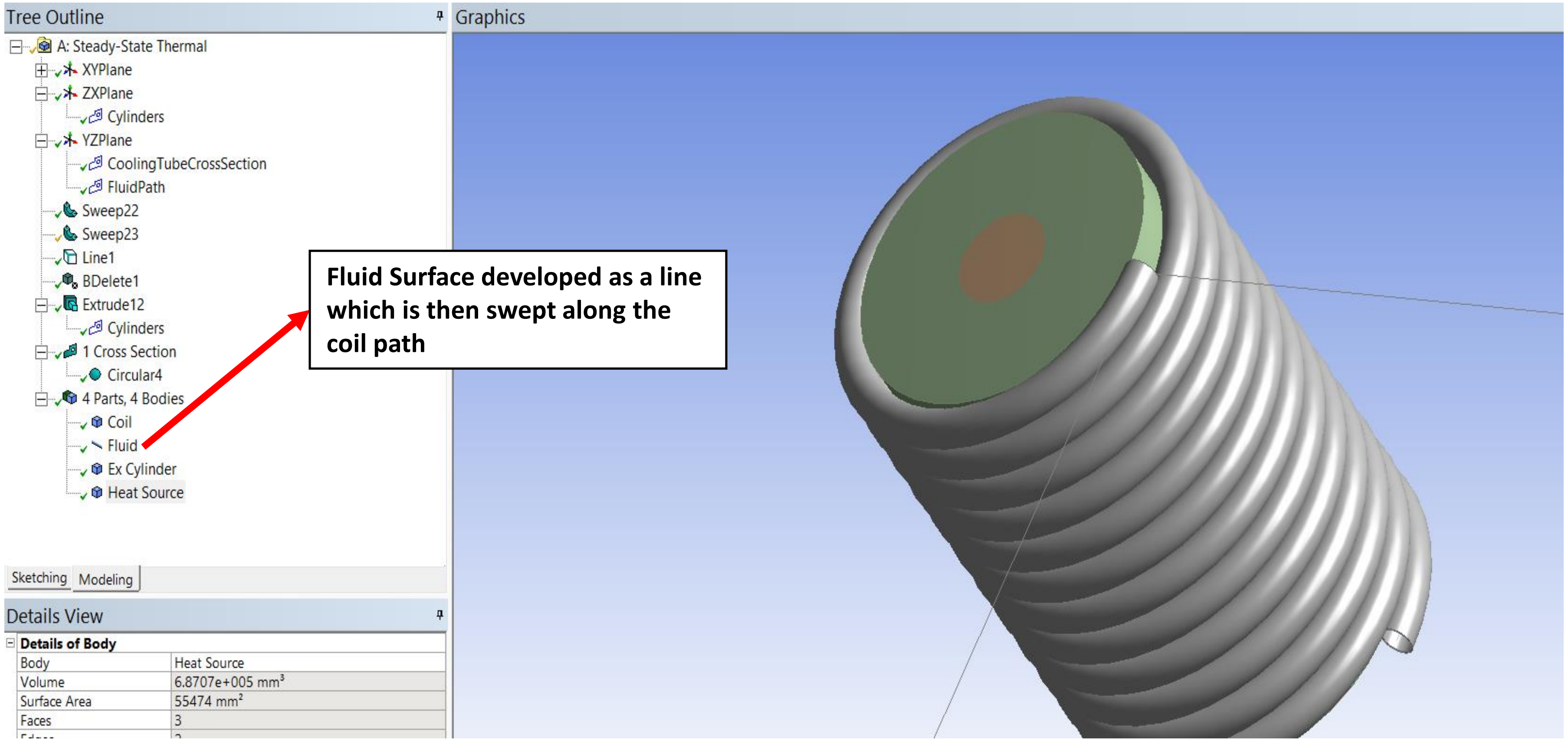
- SimuTech developed the CAD geometry of the problem
- The Material used for the cylinder is Structural Steel
- The cooling agent is considered to be water



Simplified Model



Simplified Model: Details of the Fluid



Model Development

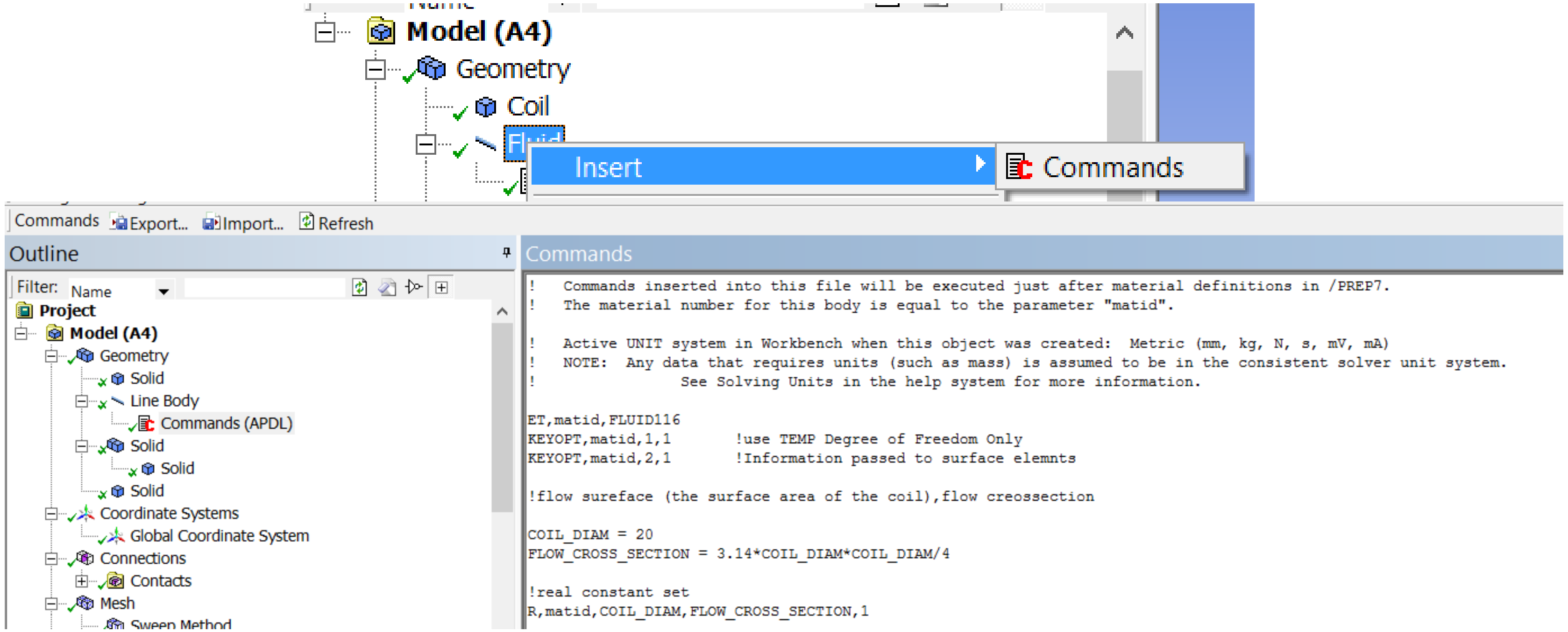
- A steady state thermal analysis is carried out with an initial inlet temperature for the fluid set to 50^0 C
- Due to lack of any conditions regarding the problem, the following assumptions were made:
  - The mass flow of water was considered to be 0.0098g/s
  - The cross section area of the cooling tube is 314 mm^2
  - The cylinder has a height of 300 mm and a diameter of 178 mm
  - A convection coefficient of 100 $w/mm^2\ ^0C$
  - Water is considered the cooling agent

Model Development

- **Fluid**
  - In order to model the mass flow of the fluid, Element Type (ET) FLUID116 is used. But once the CAD geometry is imported in Mechanical, WB will assume the line elements (Fluid) are beams. This is overwritten with a command Snippet
  - The FLUID116 element must be explicitly defined along with appropriate KEYOPTIONS
- **Heat Source and External Cylinder**
  - Solid ET SOLID70 with thermal degrees of freedom is used
- **Contact Elements**
  - In order to simulate the heat transfer between the Heat Source – External Cylinder and External Cylinder – Cooling Coil, surface to surface ET CONTA174 and TARGE170 are used. Due to the limitations on the geometry pinball radius of around 4 mm is used to define the surfaces of the coil and external cylinder
- **Convection Heat Transfer**
  - In order to simulate the heat transfer between the fluid and the internal surface of the coil via convection, surface ET SURF152 are placed on the inner surface of the coil

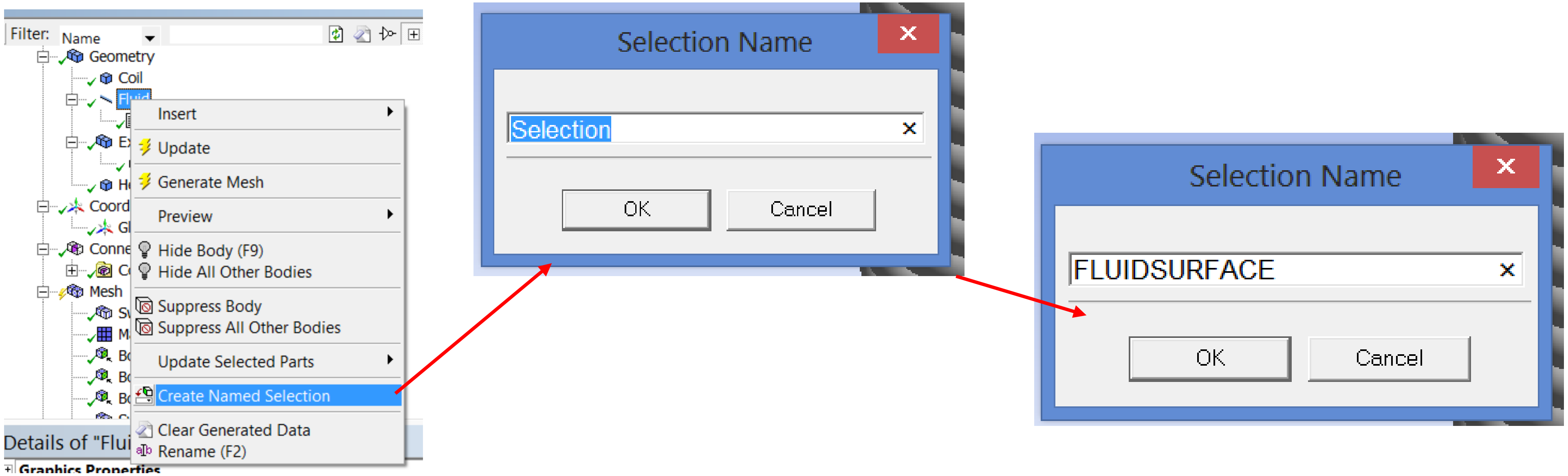
Command Snippets - Fluid

- **Fluid Geometry**
  - Right click on the Fluid geometry and insert commands
  - Unless more than one ET is used, the material identification number, the parameter ‘**matid**’ is the same as the element type number and the real constant set number



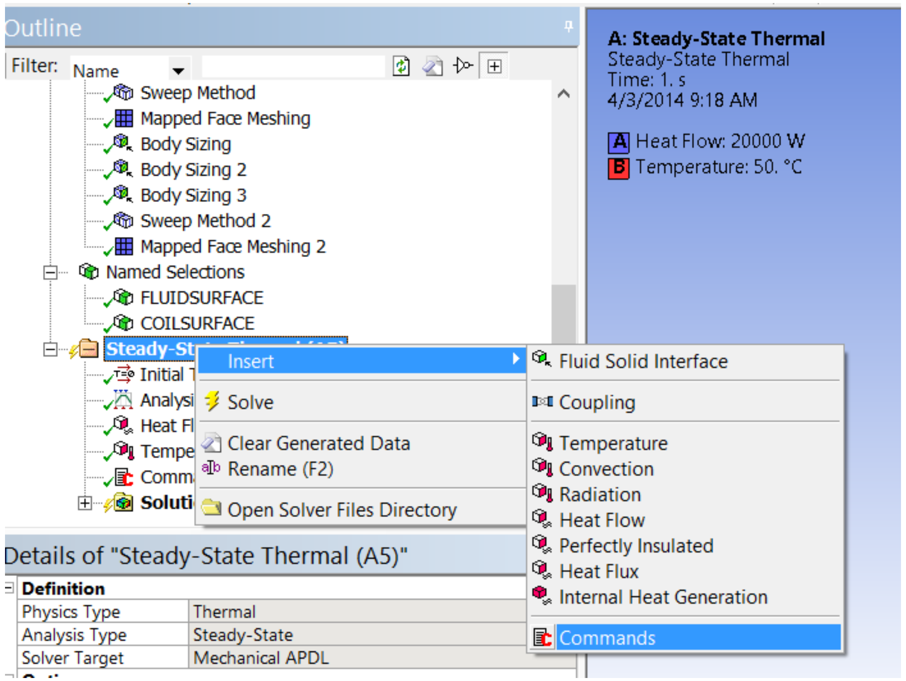
Command Snippets – Named Components

- **Named Components for Fluid Elements**
  - The ANSYS command used to create the surface elements requires named components of the convection surfaces (internal surface of coil) and named components of the Fluid Elements
  - Highlight the fluid geometry and internal face of the coil, click create component and name the resulting components FLUID SURFACE and COIL SURFACE respectfully

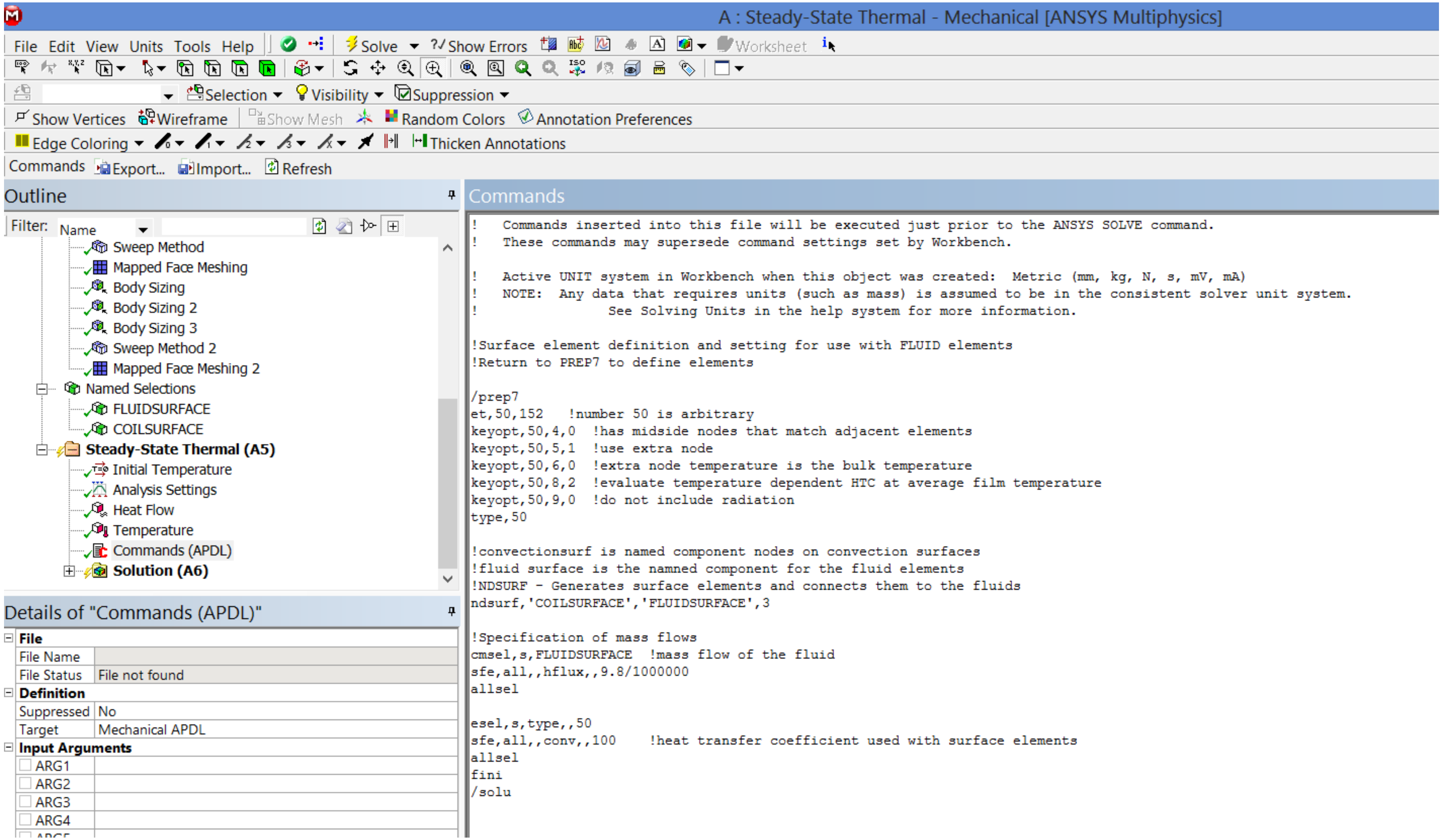


Command Snippets – Solution

- **Snippets for Simulation**
  - This is where the mass flow rate will be specified and the surface elements on the convection surfaces created and connected to the fluid elements

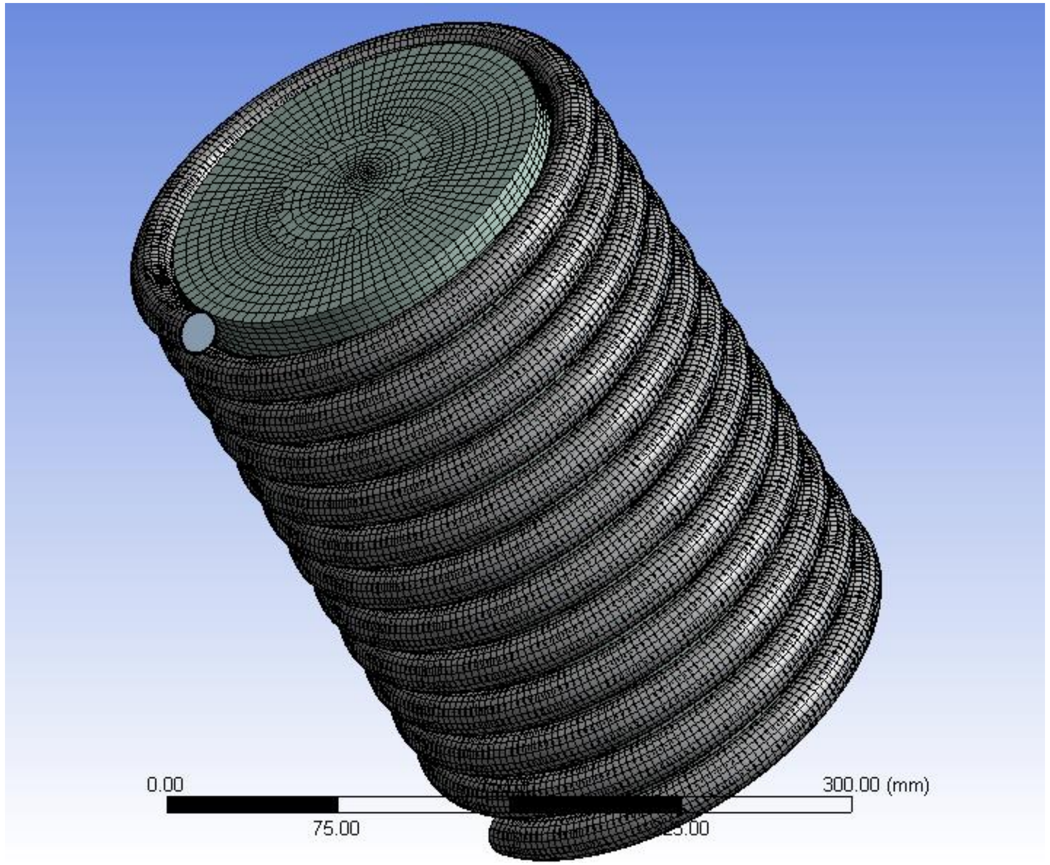


Command Snippets – Solution



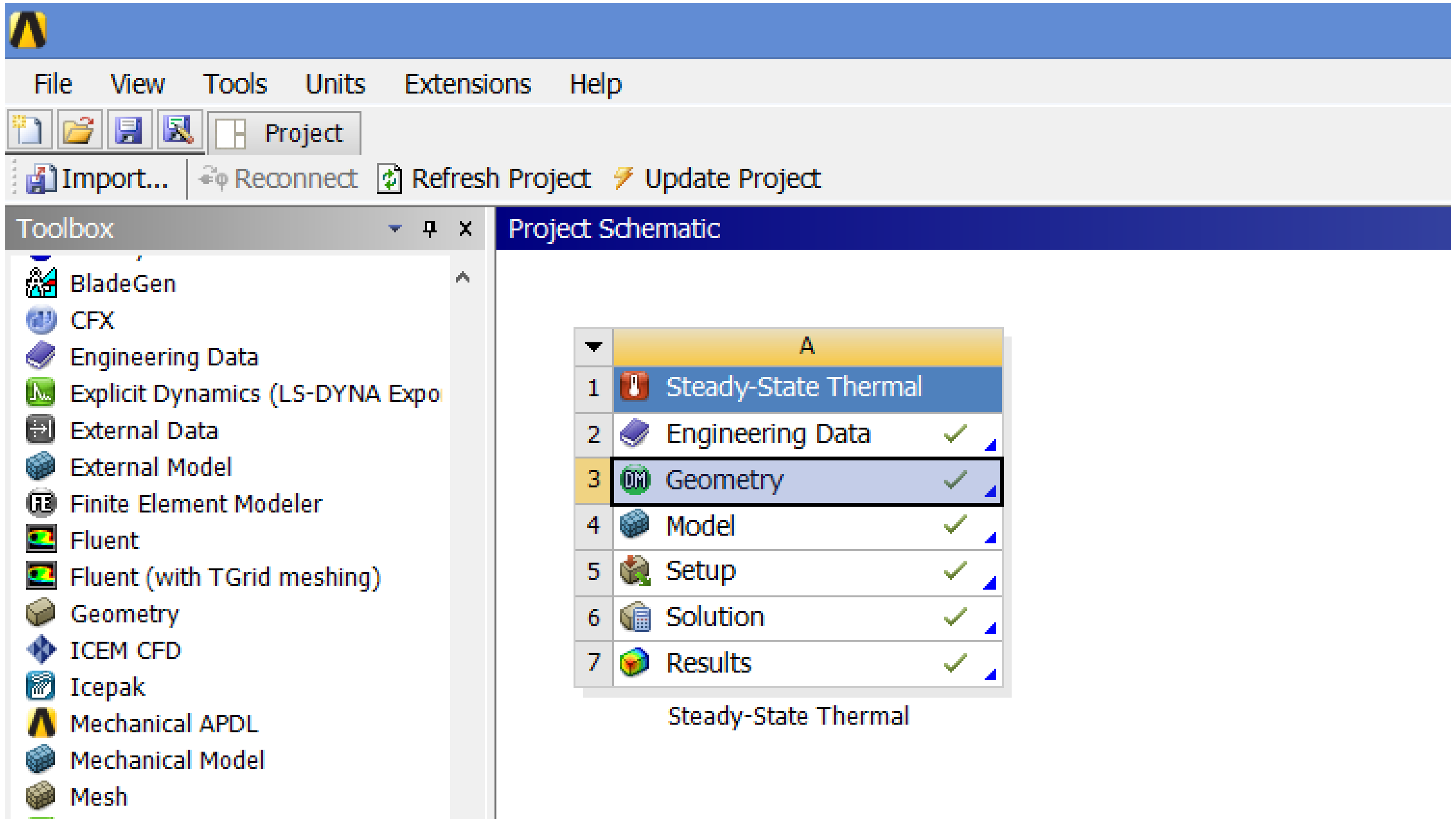
Full Model

- The FE Mesh is shown below

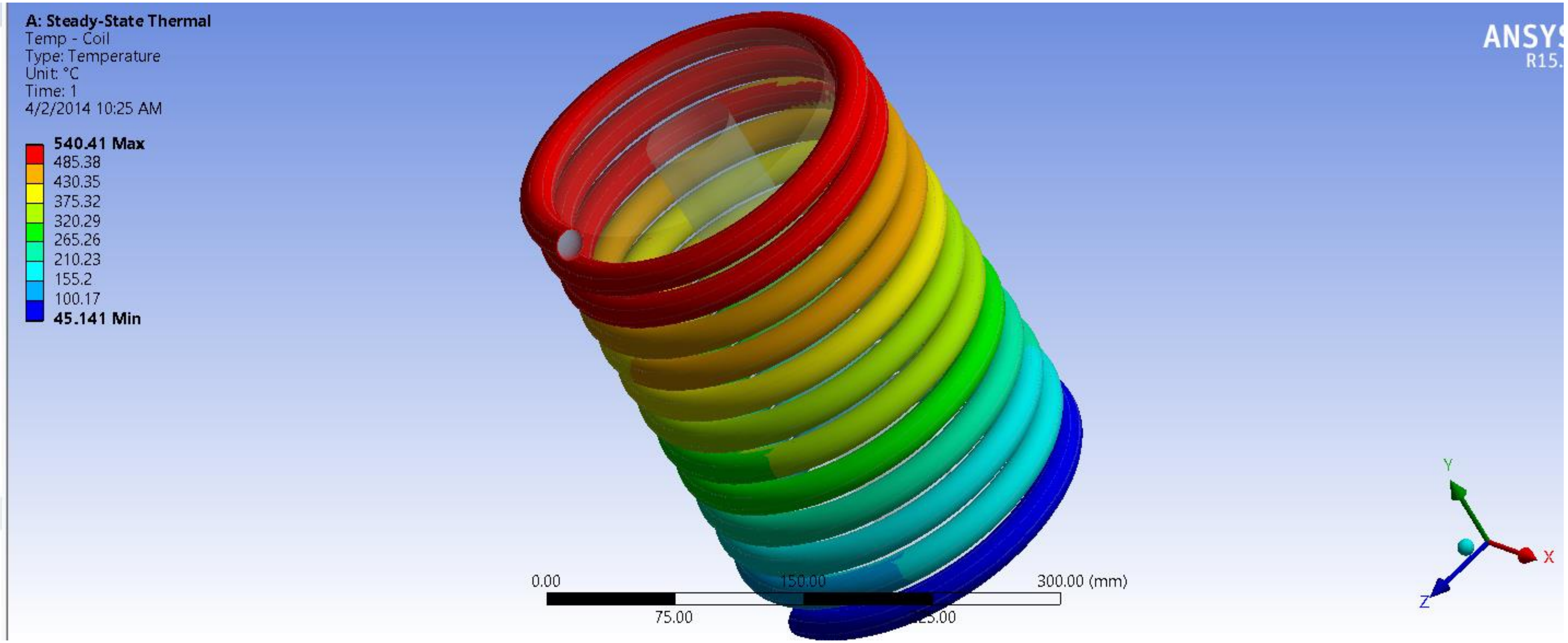


Model Development

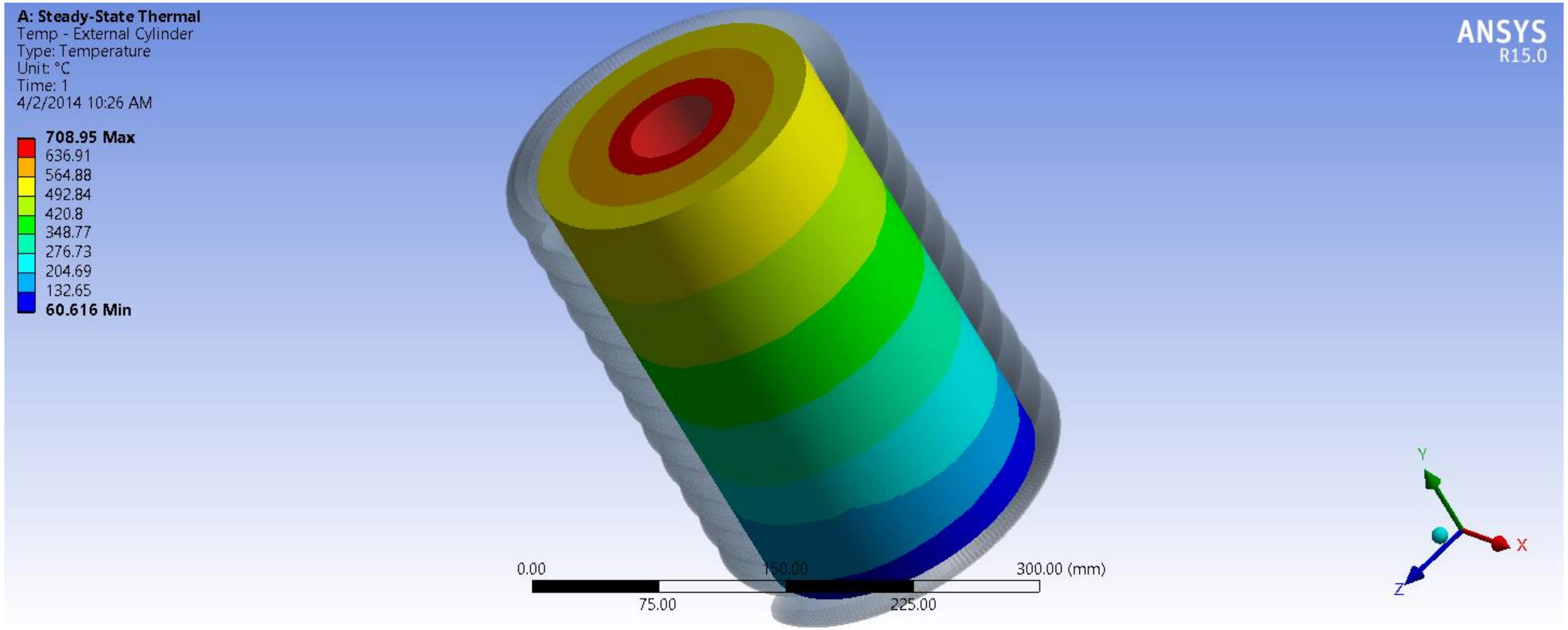
- Outline of the model



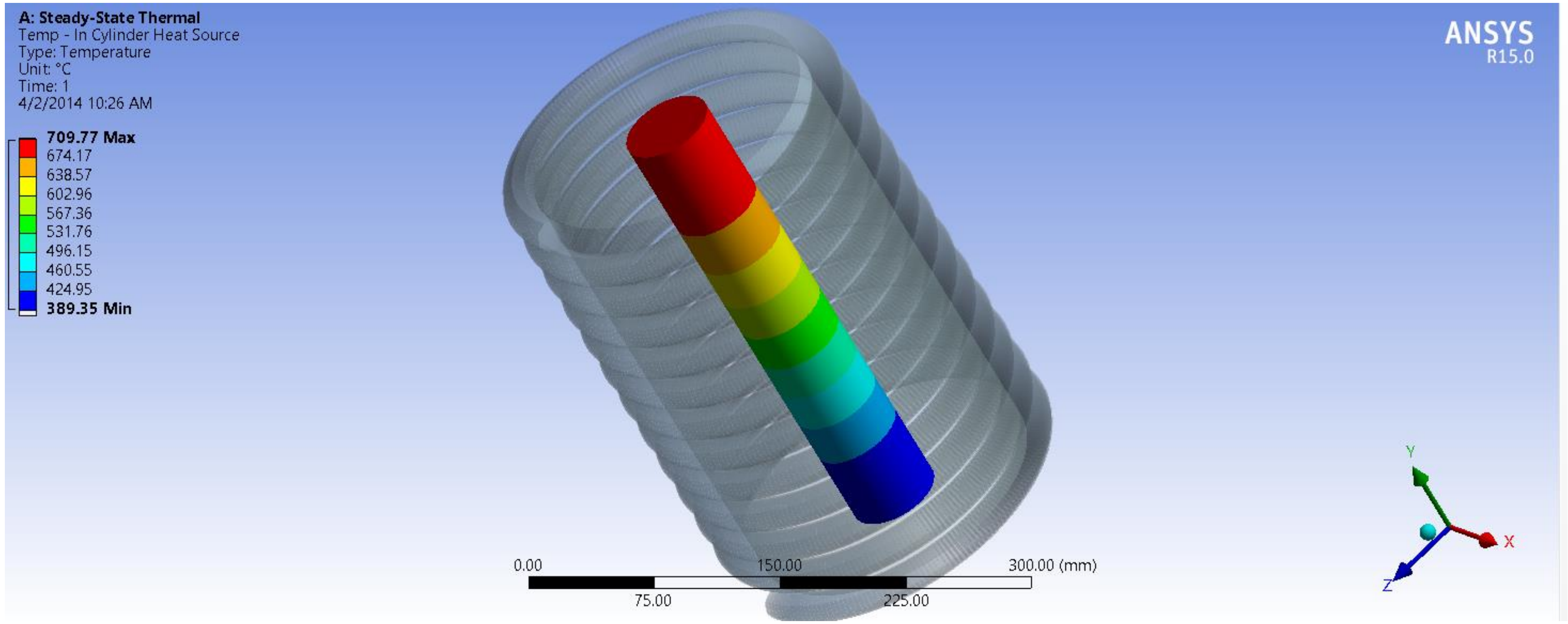
Temperature Distribution: Coil



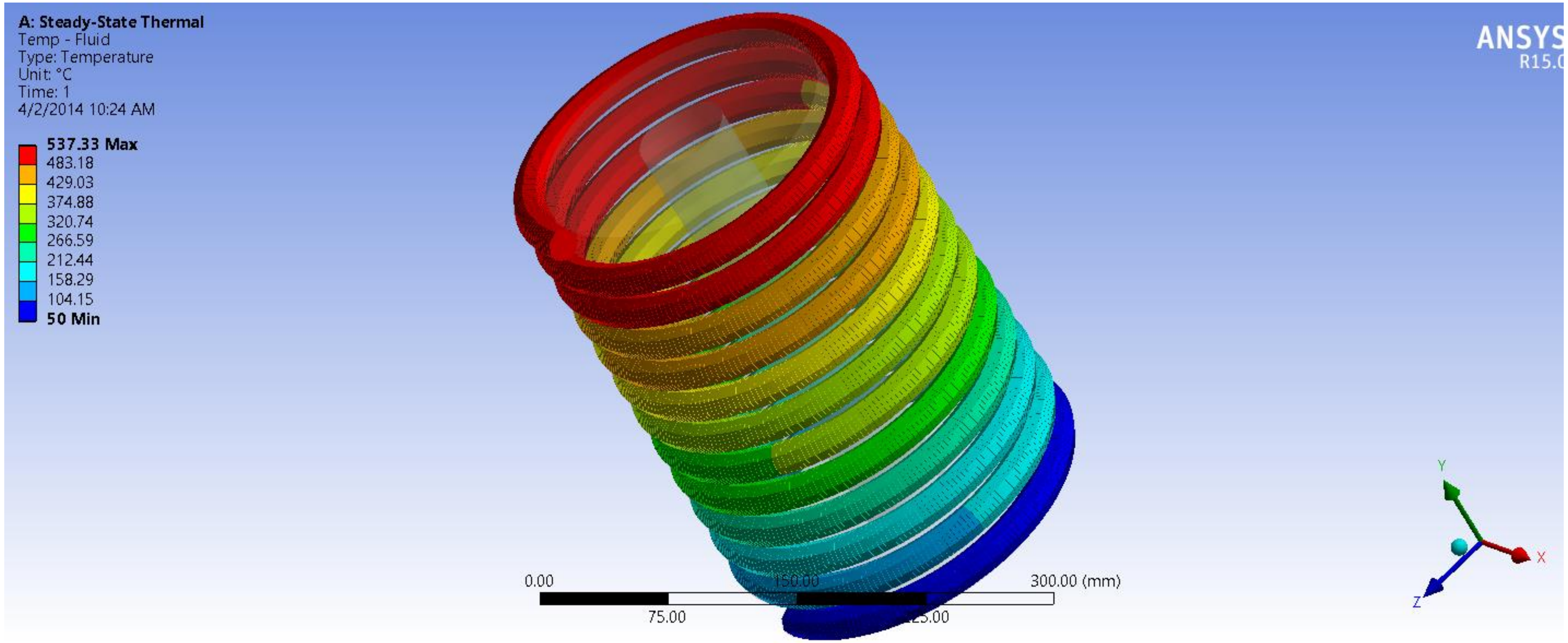
Temperature Distribution: External Cylinder



Temperature Distribution: Internal Cylinder



Temperature Distribution: Fluid



Conclusions

- This initial model was prepared in order to get a glimpse of how the cooling goes
- A more detailed model can be developed with all the physical conditions
- Parametric studies on the model can be done. This may involve different variations especially on the geometric parameters namely; the tube cross section area, mass flow rate, tube number of coils etc
- Optimization loops can also be set up with output parameters being the target temperature on the cylinder
- Convection could also be considered between the coil, cylinder and the environment