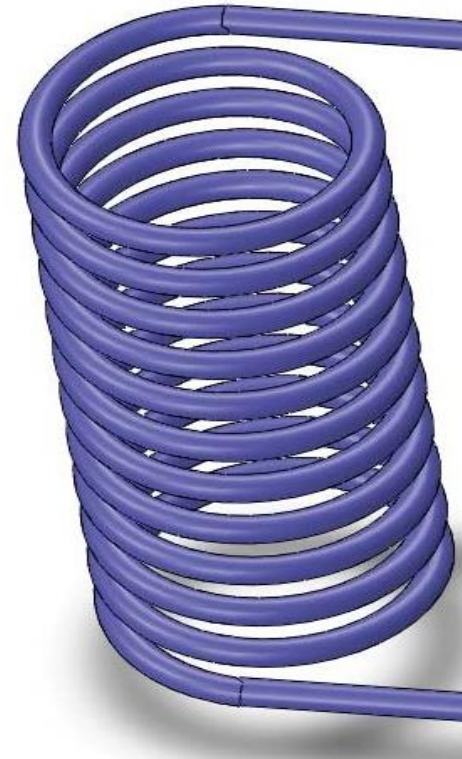


# Intro

- 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

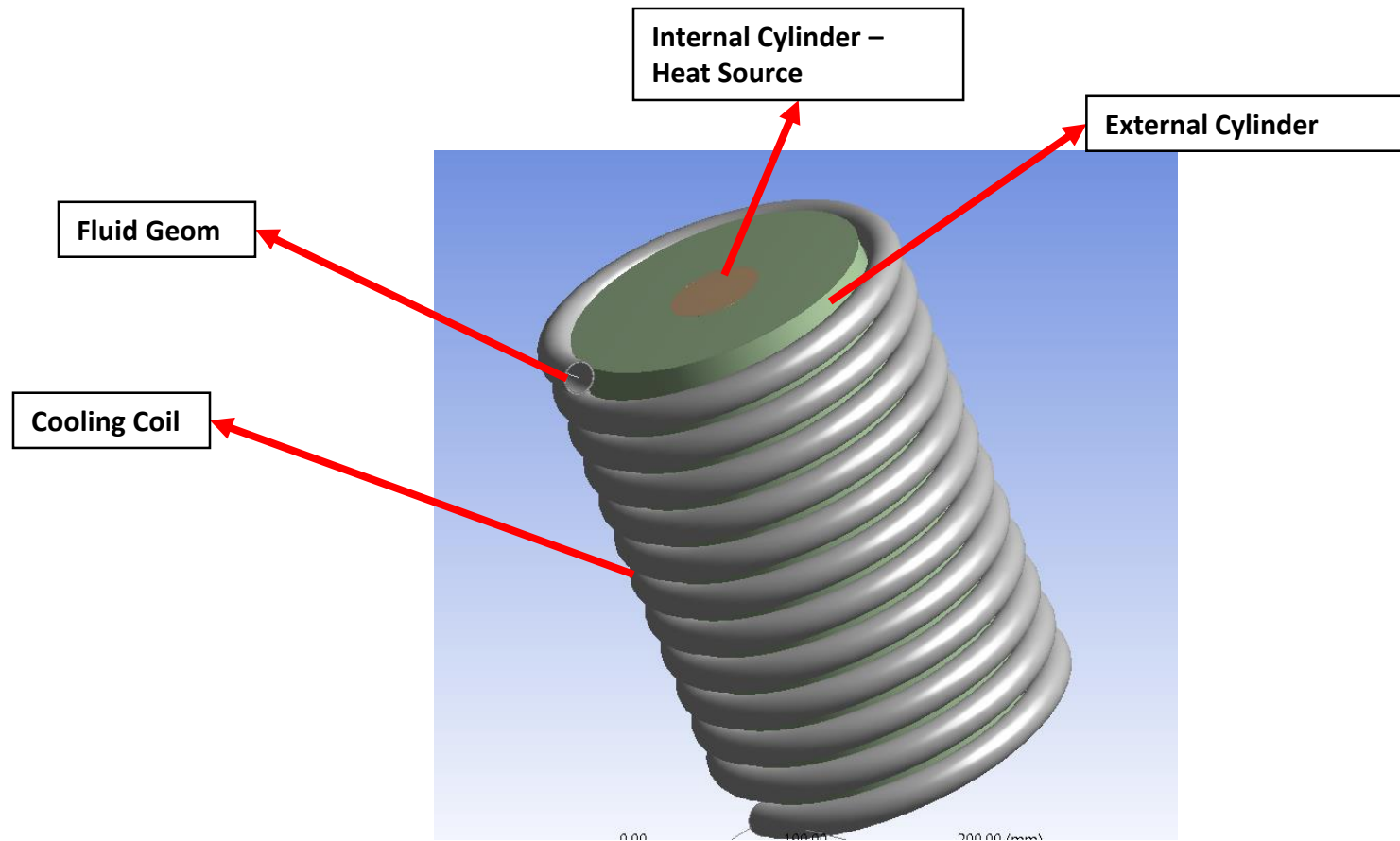
# Initial CAD Geometry

- 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

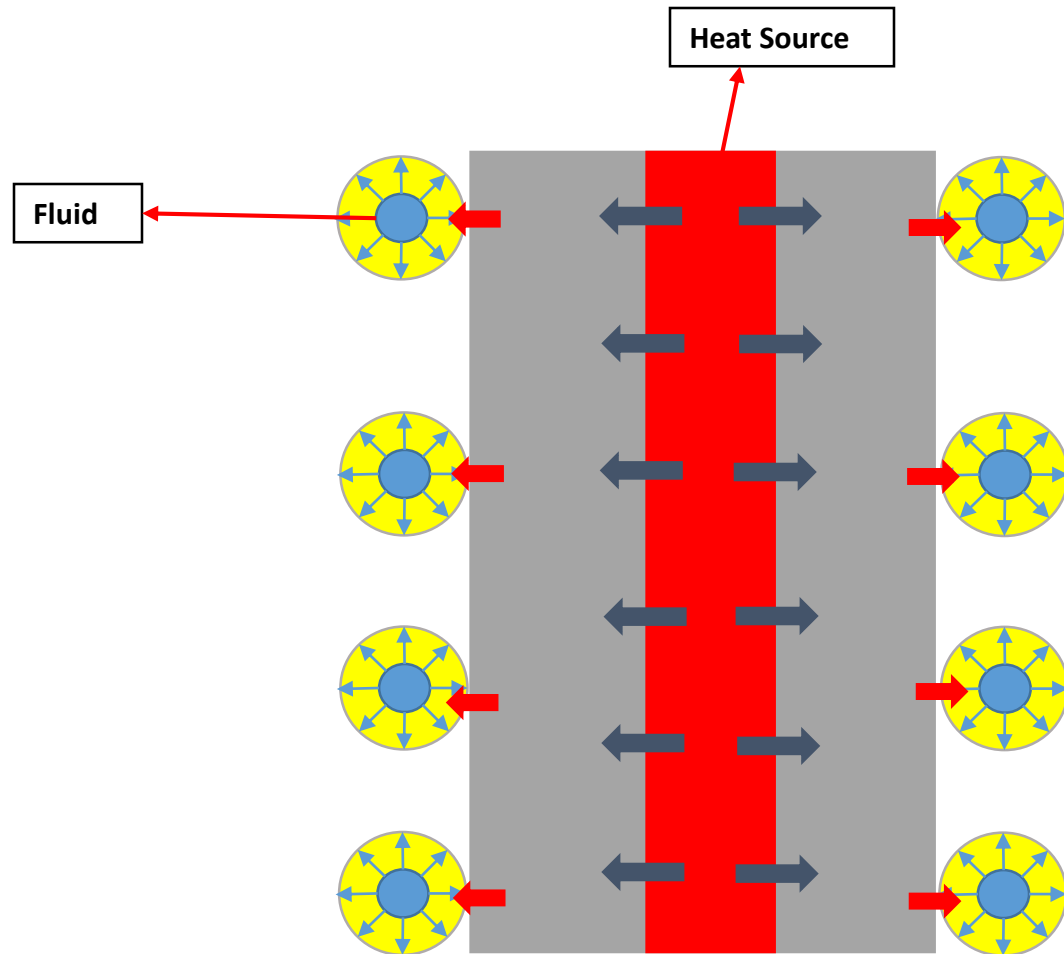


# Simplified Model

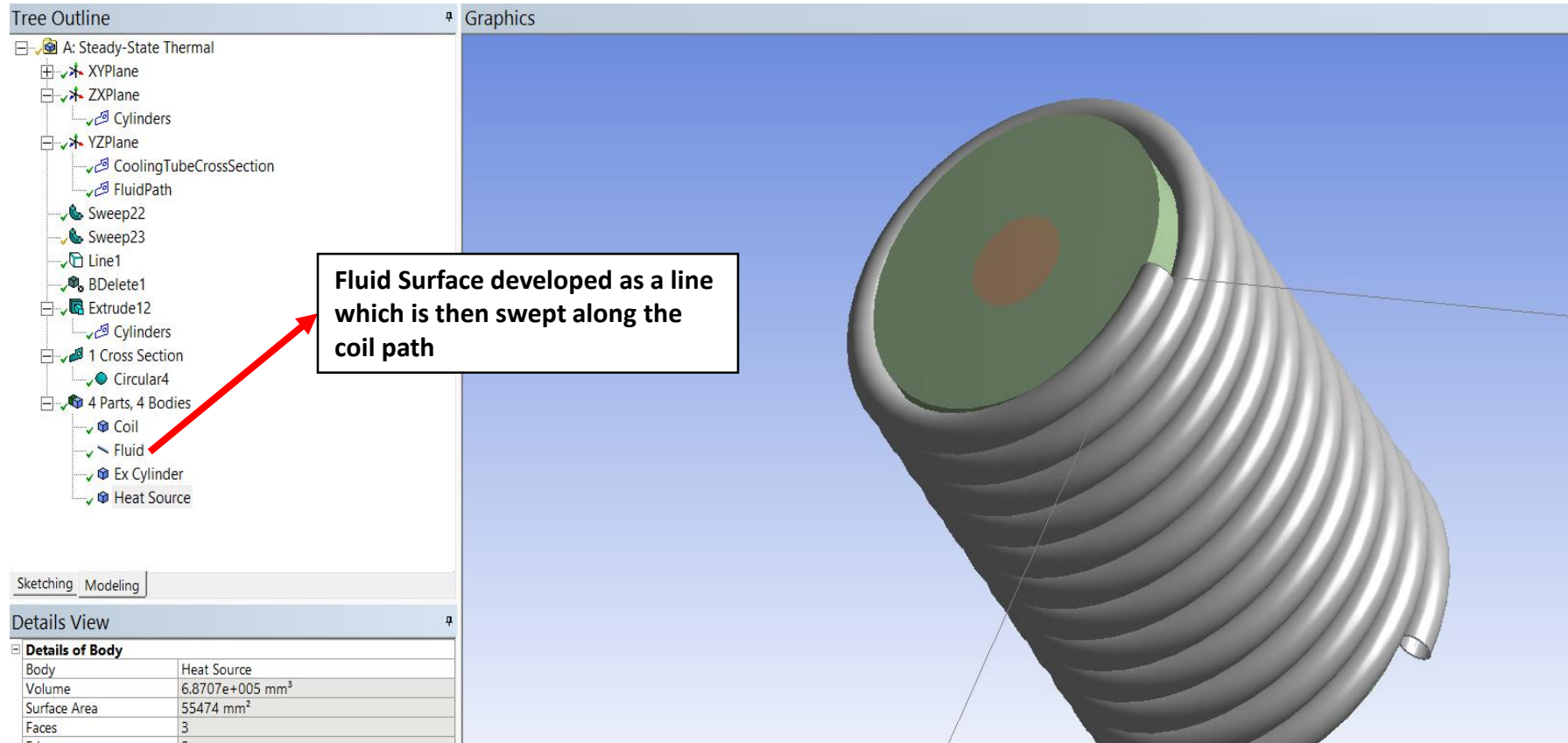
- SimuTech developed the CAD geometry of the problem
- The Material used for the cylinder is Structural Steel
- The cooling agent is considered to be water



# 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<sup>0</sup> 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<sup>2</sup>
  - The cylinder has a height of 300 mm and a diameter of 178 mm
  - A convection coefficient of 100 w/mm<sup>2</sup>°C
  - Water is considered the cooling agent

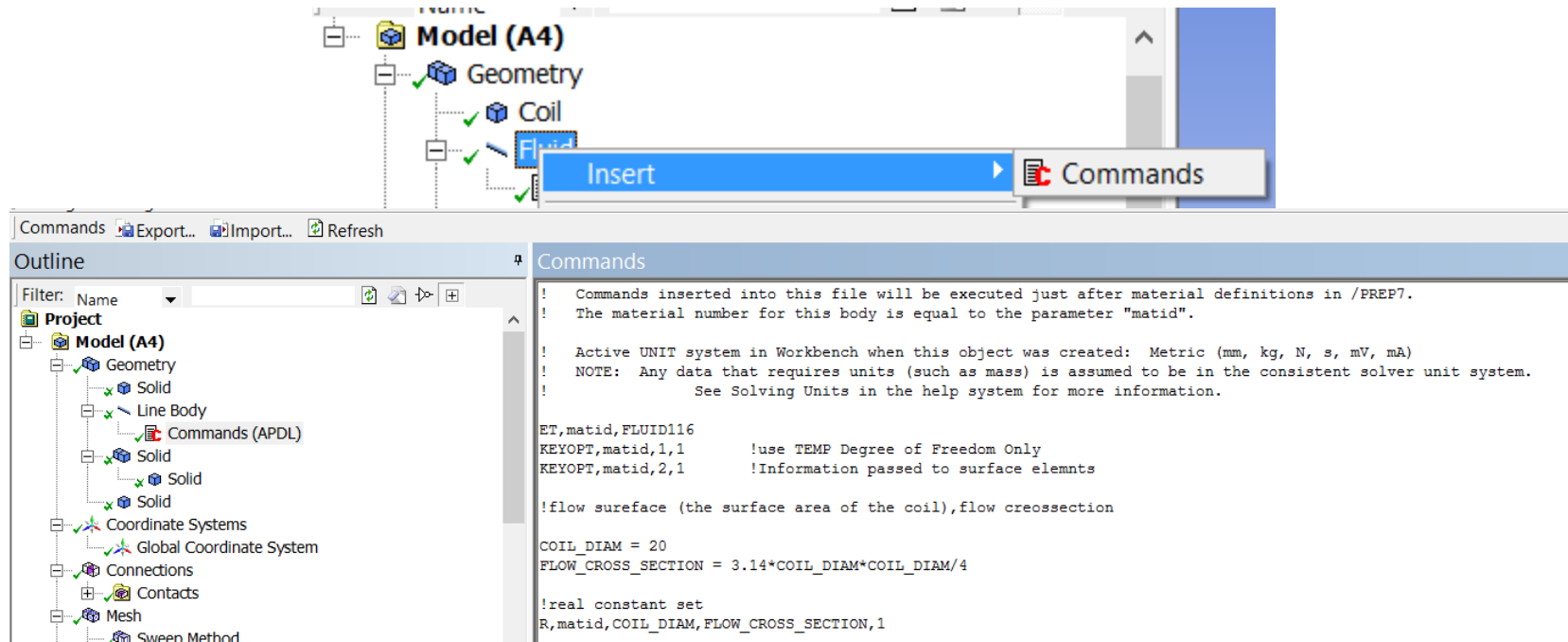
# Model Development

- **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

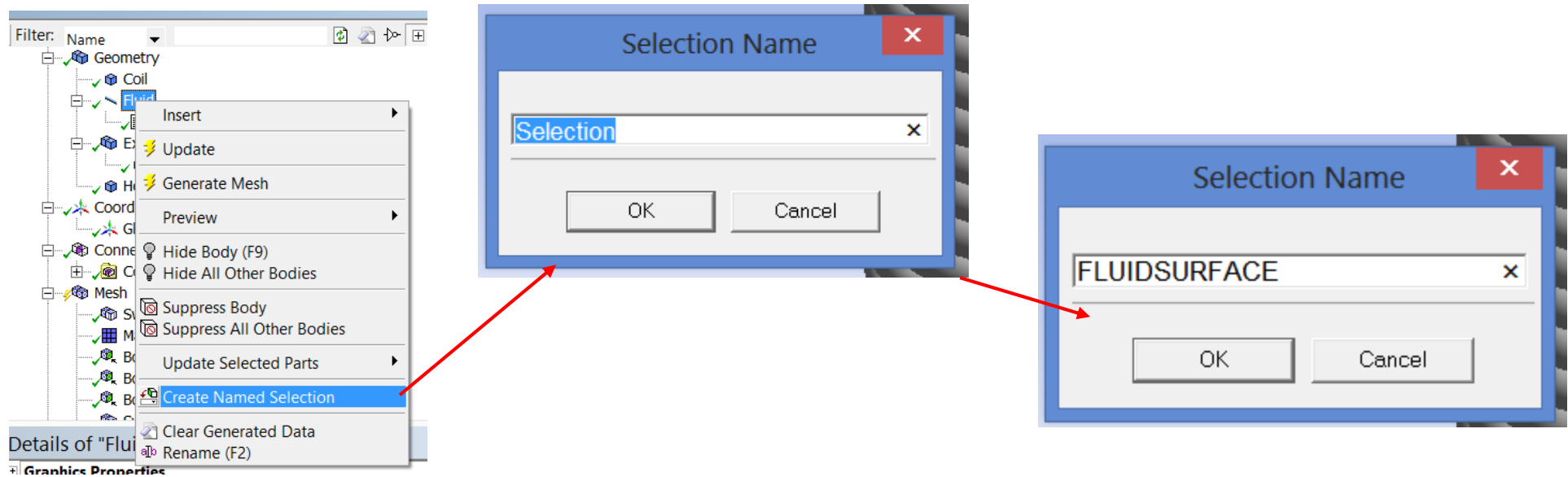




# Command Snippets – Named Components

- **Named Components for Fluid Elements**

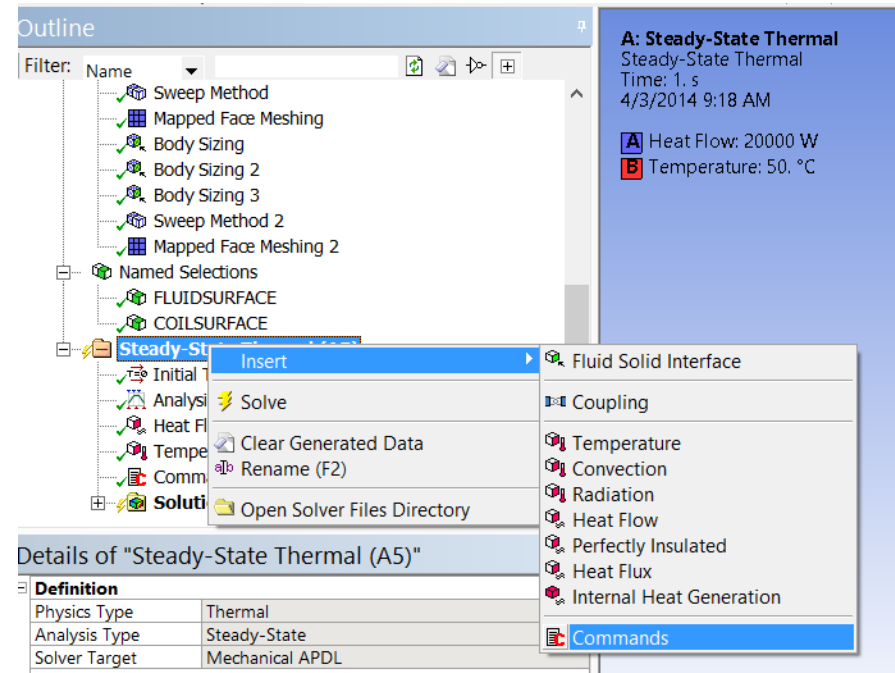
- The ANSYS command used to create the surface elements requires named components of the convection surfaces (internal surface of coil) and named components of the Fluid Elements
- Highlight the fluid geometry and internal face of the coil, click create component and name the resulting components FLUID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

A : Steady-State Thermal - Mechanical [ANSYS Multiphysics]

File Edit View Units Tools Help | Solve Show Errors Worksheet

Selection Visibility Suppression

Show Vertices Wireframe Show Mesh Random Colors Annotation Preferences

Edge Coloring Thicken Annotations

Commands Export... Import... Refresh

**Outline**

Filter: Name

- [-] Sweep Method
  - [+] Mapped Face Meshing
  - [+] Body Sizing
  - [+] Body Sizing 2
  - [+] Body Sizing 3
  - [+] Sweep Method 2
  - [+] Mapped Face Meshing 2
- [+] Named Selections
  - [+] FLUIDSURFACE
  - [+] COILSURFACE
- [+] **Steady-State Thermal (A5)**
  - [+] Initial Temperature
  - [+] Analysis Settings
  - [+] Heat Flow
  - [+] Temperature
  - [+] Commands (APDL)
  - [+] **Solution (A6)**

**Commands**

```

! Commands inserted into this file will be executed just prior to the ANSYS SOLVE command.
! These commands may supersede command settings set by Workbench.

! Active UNIT system in Workbench when this object was created: Metric (mm, kg, N, s, mV, mA)
! NOTE: Any data that requires units (such as mass) is assumed to be in the consistent solver unit system.
! See Solving Units in the help system for more information.

!Surface element definition and setting for use with FLUID elements
!Return to PREP7 to define elements

/prep7
et,50,152 !number 50 is arbitrary
keyopt,50,4,0 !has midside nodes that match adjacent elements
keyopt,50,5,1 !use extra node
keyopt,50,6,0 !extra node temperature is the bulk temperature
keyopt,50,8,2 !evaluate temperature dependent HTC at average film temperature
keyopt,50,9,0 !do not include radiation
type,50

!convection surf is named component nodes on convection surfaces
!fluid surface is the named component for the fluid elements
!NDSURF - Generates surface elements and connects them to the fluids
ndsurf,'COILSURFACE','FLUIDSURFACE',3

!Specification of mass flows
cme1,s,FLUIDSURFACE !mass flow of the fluid
sfe,all,,hflux,,9.8/1000000
allsel

esel,s,type,,50
sfe,all,,conv,,100 !heat transfer coefficient used with surface elements
allsel
fini
/solu

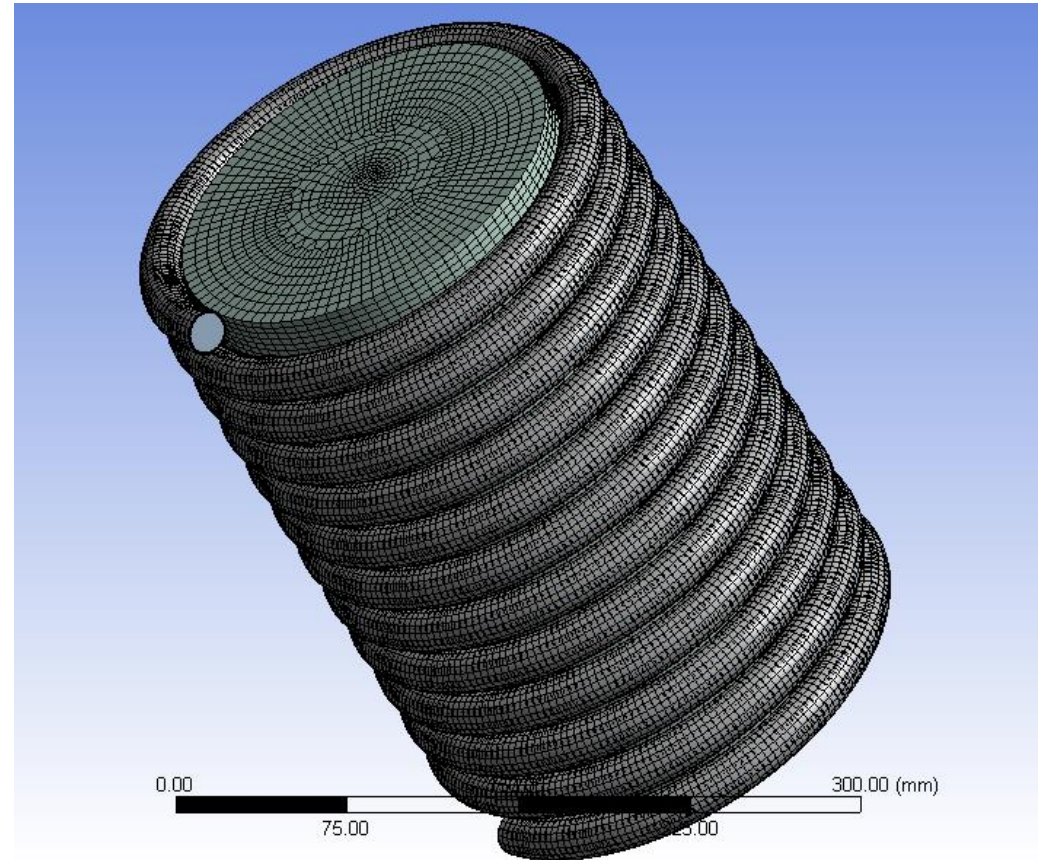
```

**Details of "Commands (APDL)"**

<b>File</b>	
File Name	
File Status	File not found
<b>Definition</b>	
Suppressed	No
Target	Mechanical APDL
<b>Input Arguments</b>	
ARG1	
ARG2	
ARG3	
ARG4	
ARG5	

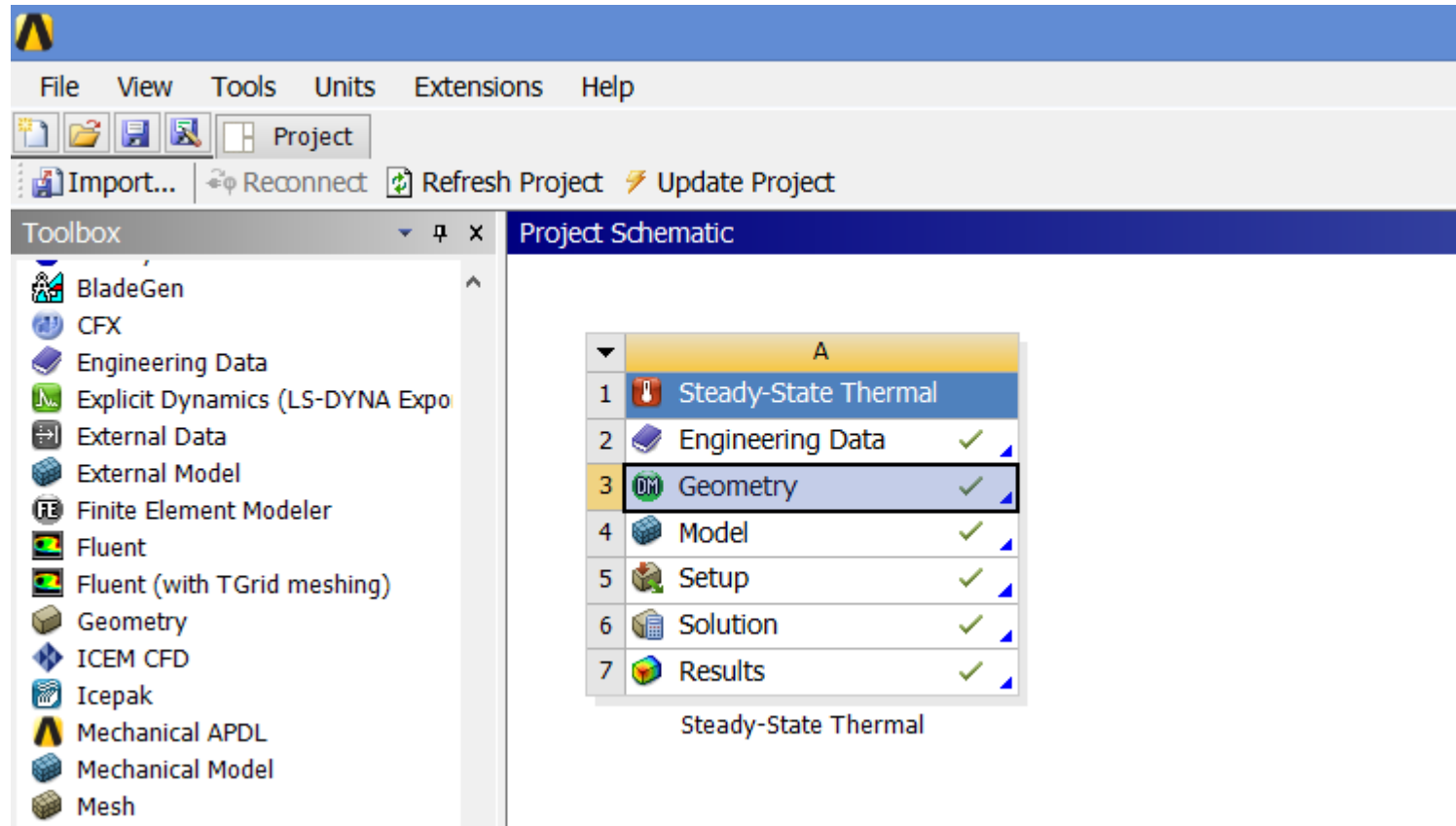
# Full Model

- The FE Mesh is shown below

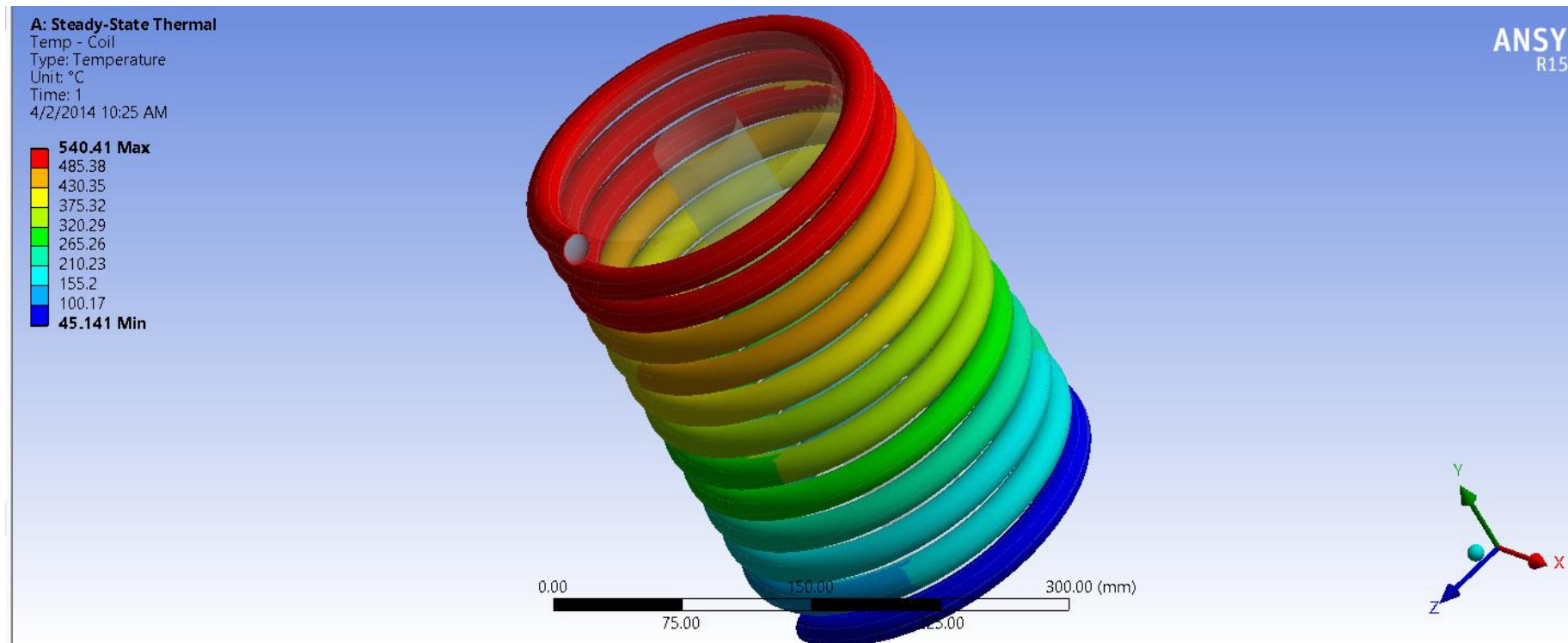


# Model Development

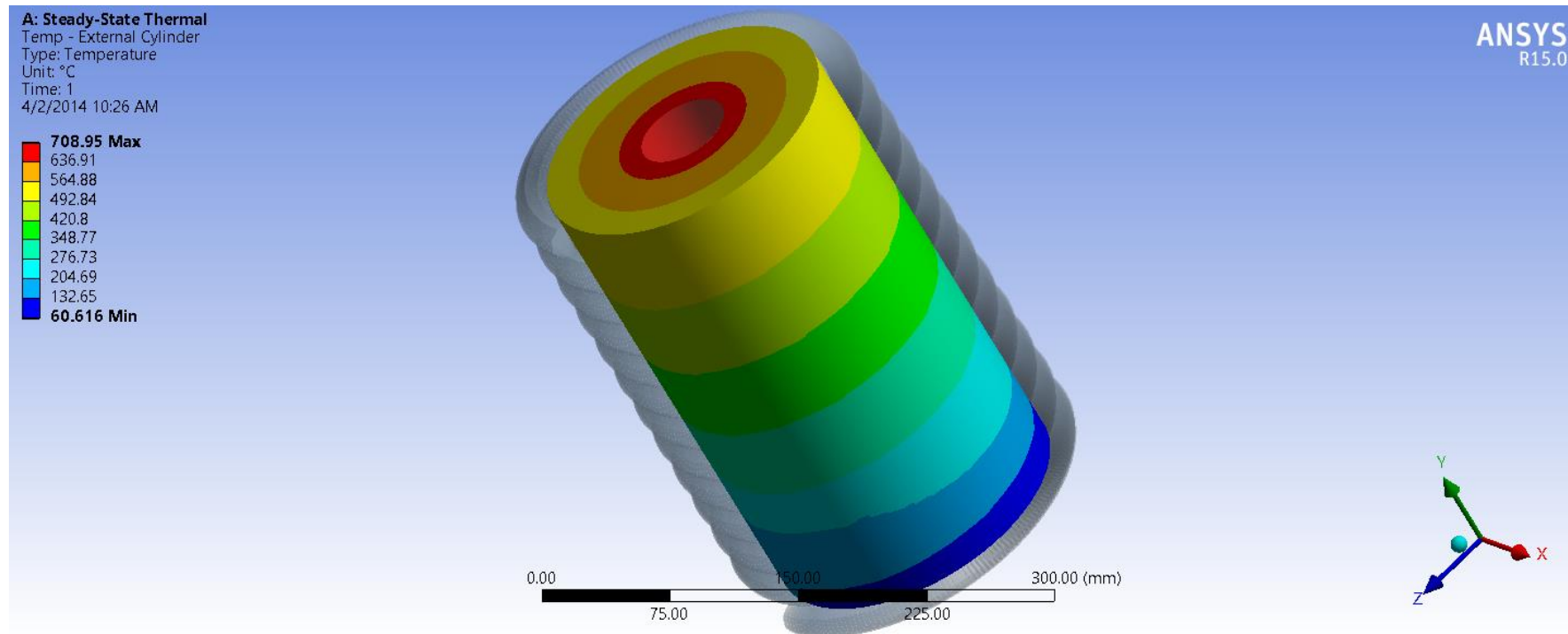
- Outline of the model



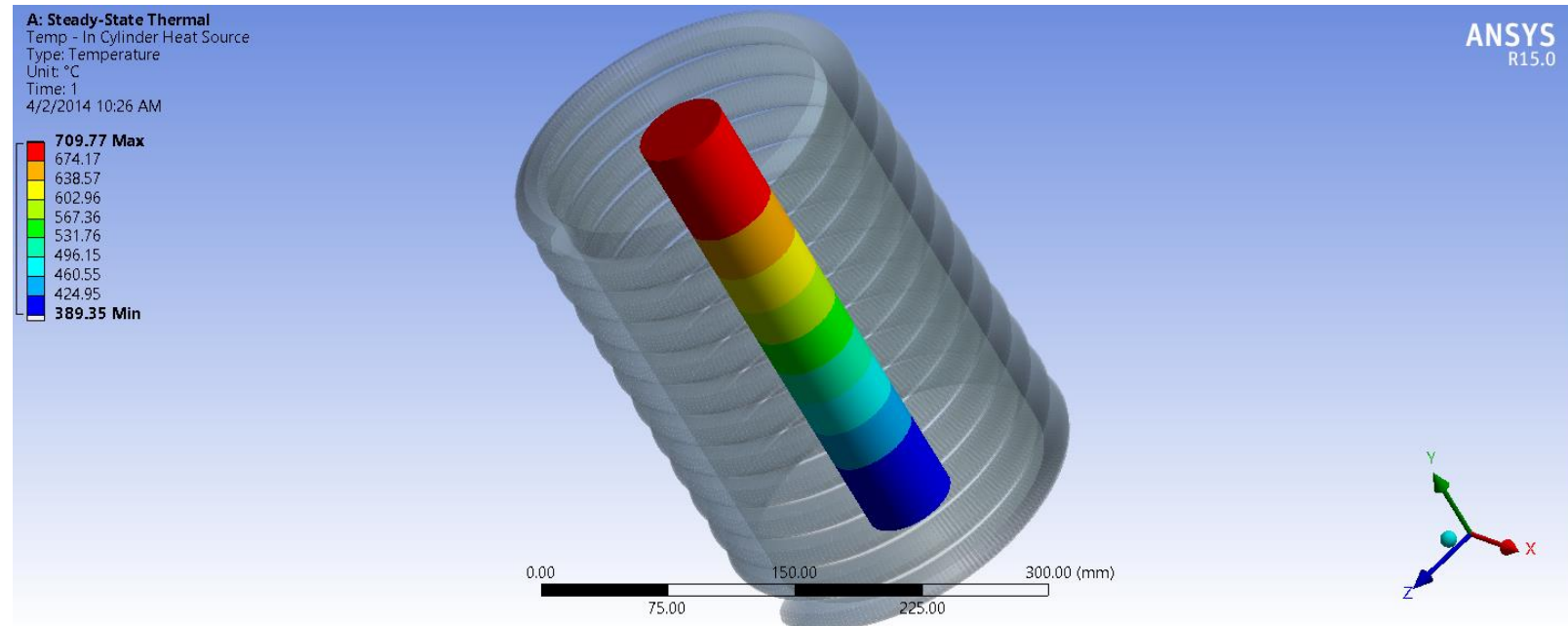
# Temperature Distribution: Coil



# Temperature Distribution: External Cylinder

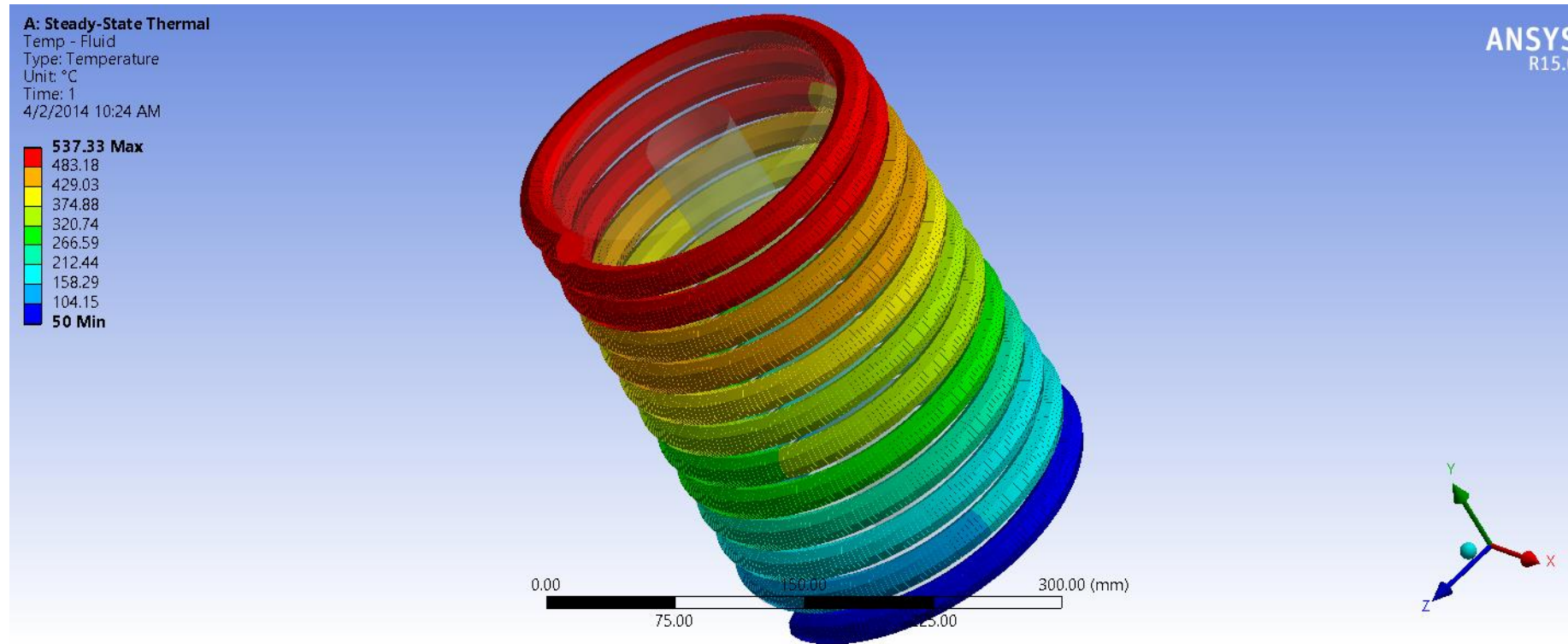


# Temperature Distribution: Internal Cylinder





# 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