MATH5004 LAB 11

Part I 3D FEM with MATLAB

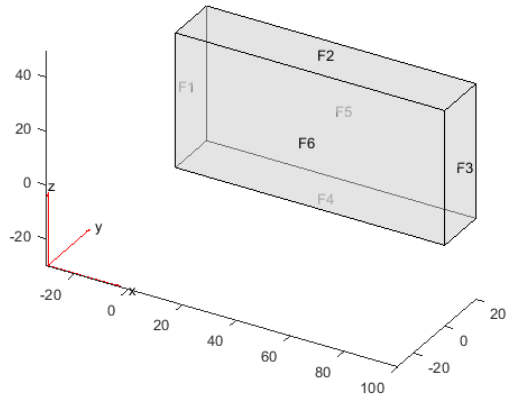
Consider the heat transfer problem described by

$$\frac{\partial u}{\partial t} - \nabla \cdot \nabla u = 0, \quad (x,y,z) \in \Omega, t \in (0,\tau)$$

subject to

$$u(x,y,z,0) = 350^{o}C$$

$$\frac{\partial u}{\partial n}(x,y,z,t) = -(1 \times 10^{-6})\, u^3 \quad (x,y,z) \in \partial\Omega.$$



As the general form of PDE in MATLAB is

$$m\frac{\partial^2 u}{\partial t^2} + d\frac{\partial u}{\partial t} - \nabla \cdot (c\nabla u) + au = f,$$

we then have coefficients $m = 0, d = 1, c = 1, a = 0$ and $f = 0$.

- Define and plot geometry

```
model = createpde();
importGeometry(model,'Block.stl');
figure(1)
pdegplot(model,'FaceLabel','on','FaceAlpha',0.5)
axis equal
```

- Define equations (PDE coefficients)

```
specifyCoefficients(model,'m',0,...
                          'd',1,...
                          'c',1,...
                          'a',0,...
                          'f',0);
```

- Define boundary condition

```
gfun = @(region,state)-state.u.^3*1e-6;
applyBoundaryCondition(model,'neumann','Face',1:model.Geometry.NumFaces,...
'g',gfun);
```

- Define initial condition

```
setInitialConditions(model,350);
```

- Mesh over the geometry

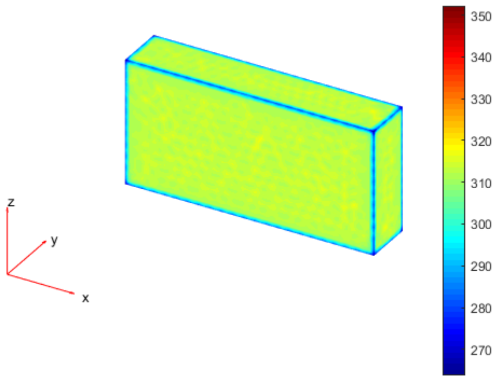
```
generateMesh(model);
```

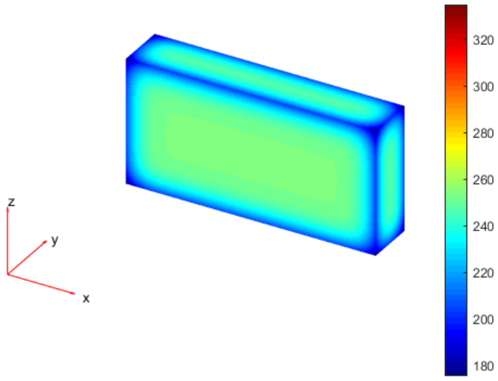
- Solve results

```
tlist = 0:20;
results = solvepde(model,tlist);
```

- Visualize results

```
figure(1)
pdeplot3D(model,'ColorMapData',results.NodalSolution(:,2))
figure (2)
pdeplot3D(model,'ColorMapData',results.NodalSolution(:,21))
```





Part II The Best Simulation Examples using ANSYS

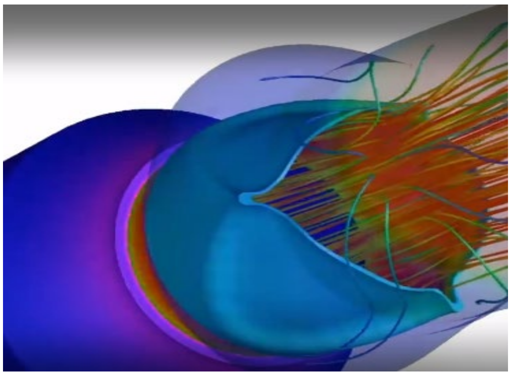
Every year the Ansys Hall of Fame (HoF) awards three academic and three commercial teams for their amazing simulation examples. These simulation examples will save lives and save millions of dollars.

In 2019, seven simulation examples have been presented at

https://www.ansys.com/blog/ansys-hof-2020

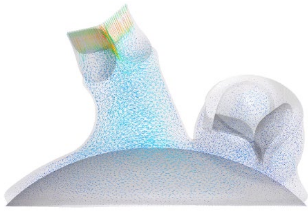
1. Fluid-Structure Interaction analysis of custom valve coupled with patient data

Ubaldo Cella, a researcher from Tor Vergata applied a computational fluid dynamics (CFD) simulation with moving wall boundary to study the mechanics of the heart and cardiovascular systems. The simulation of moving walls and fluid–structure interactions of a cardiovascular system was used to assess custom valves that are coupled to individual patient data. By bringing these simulations into the clinic, Cella will help doctors save lives.



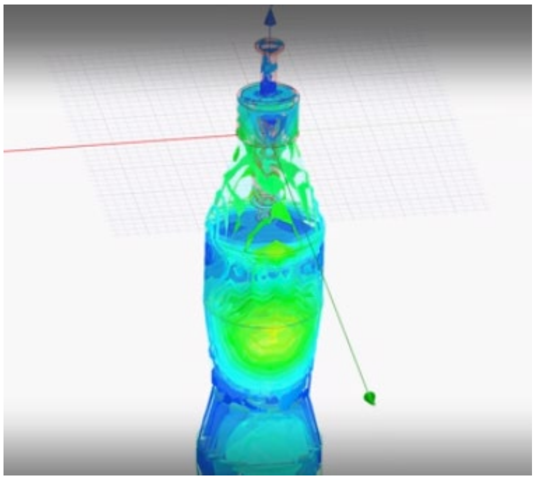
2. Valves and blood flow associate with an artificial ventricle

Giulia Luraghi, a PhD engineer from Politecnico di Milano, used Fluent and Ansys LS-DYNA to simulate the valves and blood flow associated with an artificial ventricle.



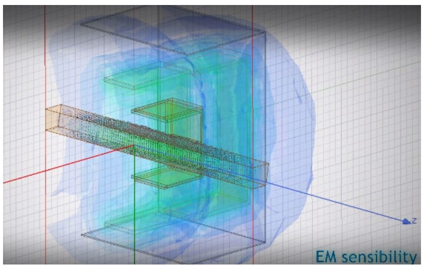
3. Dielectric materials in plastic

Due to their plastic containers, conventional radar cannot detect weapons, Landmines thus continue to be a danger to military personnel and civilians long after wars end.
Sebastian Celis and Li Zhang, graduate students, and Mohamed Farhat, Ph.D , researchers from King Abdullah University of Science and Technology used Ansys HFSS and machine learning to detect plastic bottles filled with dielectric materials that mimic how explosives react to high frequencies. With this technology, the researchers can save countless lives around war torn areas.



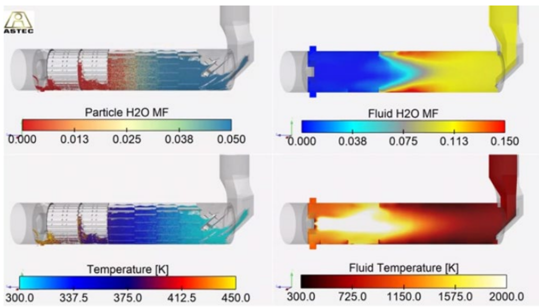
4. Copper Ingot Cavity

Ingot cavities can cause significant production shutdowns from the copper ingots. Mario Gregorio Angeloni at Druids Process Technology developed an electromagnetic detector to find copper ingot cavity. The simulation displays the ability to detect small cavities in high temperature copper ingots.

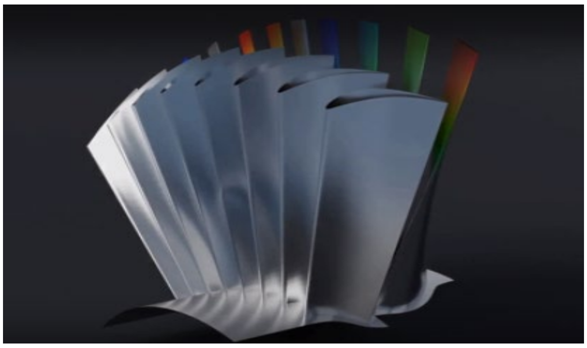


5. Mass transfer between aggregate particles

Andrew Hobbs, an engineer from Astec worked on hot asphalt aggregate dryers. Traditionally, half of the energy input of these dryers goes into the phase change. Hobbs developed software utilizing Fluent to assess the momentum and energy exchange between the fluid phase. The simulation displays the mass transfer between the aggregate particles. The software. With this tool, an efficient dryer reduces costs and emissions.



6. Blade flutter.
   Flutter describes the self-excited vibrations caused by fluid-structure interactions in rotating machines. These vibrations can cause turbomachinery to experience blade loss, damage and failure.



Tomáš Peterka, an engineer from NUM solutions, worked on flutter. He developed the FLTR tool within Ansys CFX to predict blade flutter. With this tool, simulation time, and pre- and post-processing time reduce by 60% and 95%, respectively.

------------------------- End of Document -------------------------