

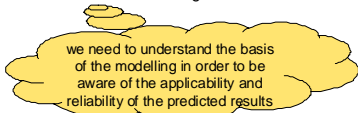







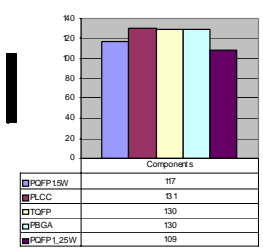


Turning the models into useful form

 <p>Turning the models into useful form</p> <ul style="list-style-type: none">▪ The ultimate goal of thermal analysis is the reduction of thermally-associated risk to the product <p style="text-align: right;"><small>Electronics KTN – Knowledge For Growth</small></p>	<p>So far we have looked in some detail at how individual heat-generating components are modelled, so we now need to turn these models into a form that is useful for projecting the thermal performance of a product. Our ultimate aim by carrying out this thermal analysis is to reduce any thermally-associated risk, so that we can identify and address issues at a very early stage in the design process.</p>
 <p>Turning the models into useful form</p> <ul style="list-style-type: none">▪ The ultimate goal of thermal analysis is the reduction of thermally-associated risk to the product▪ Analysis and simulation techniques can provide fast, accurate and cost-effective ways of predicting thermal behaviour of the product<ul style="list-style-type: none">▪ models of the heat-generating elements are important▪ may be imported from external data or integrated  <p style="text-align: right;"><small>Electronics KTN – Knowledge For Growth</small></p>	<p>To help us in the task of predicting the thermal behaviour of a product, a number of analysis and simulation techniques are available, with varying accuracy, speed and cost.</p> <p>All of these use models of the heat-generating elements, whether detailed or compact, and the accuracy and validity of these models is important. The models themselves may be imported from external data, or calculated from scratch within the tool being used for the complete system.</p> <p>Irrespective of whether we build or buy the model, we need to understand its basis, so that we are fully aware of the grounds on which our predictions are made, and realise the limits of their validity and reliability.</p>
 <p>Turning the models into useful form</p> <ul style="list-style-type: none">▪ The ultimate goal of thermal analysis is the reduction of thermally-associated risk to the product▪ Analysis and simulation techniques can provide fast, accurate and cost-effective ways of predicting thermal behaviour of the product<ul style="list-style-type: none">▪ models of the heat-generating elements are important▪ may be imported from external data or integrated▪ Many different software tools are available, varying widely in capability, ease of use and cost▪ Using (the right) tools can be particularly effective when this is done early in the design process <p style="text-align: right;"><small>Electronics KTN – Knowledge For Growth</small></p>	<p>A range of different software tools is available, varying widely in capability, ease of use, and cost. As with other types of simulation, there are many trade-offs, and the selection will depend on the budget available and also the level of expertise and the time available for thermal modelling.</p> <p>Whichever tools are chosen, using them early in the design process is particularly effective, as this reduces the costs associated with design change.</p>

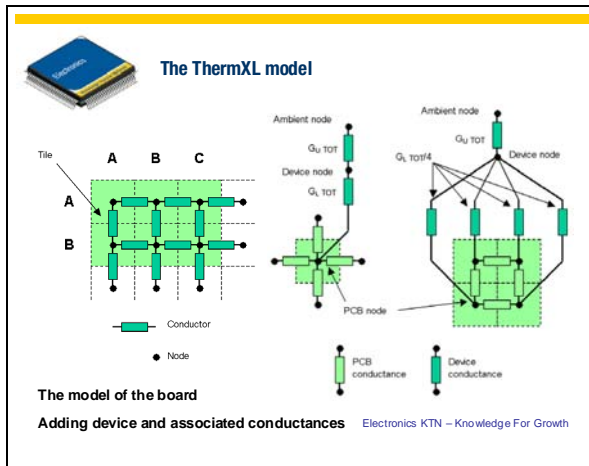
Turning the models into useful form

 <p>Analysis and simulation techniques</p> <ul style="list-style-type: none"> ▪ Hand calculation <ul style="list-style-type: none"> ▪ based on design power and package specifications ▪ Spreadsheets <ul style="list-style-type: none"> ▪ can be more than just an aid to calculation <p style="text-align: center;">tends to focus on thermal resistance models</p> <p style="text-align: center;">work on the basis of thermal resistance models of both conduction and convection</p> <p style="text-align: right; font-size: small;">Electronics KTN – Knowledge For Growth</p>	<p>Even if we have the most sophisticated tools available, using hand calculation to scope the problem is always recommended, not least as a “sanity check” to check that the model has not been built on unrealistic foundations.</p> <p>Based on the design power and information on the package specifications, it is possible to identify problem areas for further investigation. Extending the hand calculations by using spreadsheets is fine for crunching numbers, and there are many numbers to crunch when we consider all the options! But in complex cases we can also employ spreadsheets to make repeated calculations where we need to estimate answers rather than explicitly calculate them.</p> <p>Using a spreadsheet helps us to extend our focus from just thermal resistance models of both conduction and convection, and carry out sensitivity analysis to examine the effect of changing our assumptions and data.</p>
 <p>Analysis and simulation techniques</p> <ul style="list-style-type: none"> ▪ Hand calculation <ul style="list-style-type: none"> ▪ based on design power and package specifications ▪ Spreadsheets <ul style="list-style-type: none"> ▪ can be more than just an aid to calculation ▪ Flow Network Modelling (FNM) <ul style="list-style-type: none"> ▪ a way of estimating air flow within structures ▪ Computational Fluid Dynamics (CFD) <p style="text-align: center;">give information about the actual flow of air and heat</p> <p style="text-align: right; font-size: small;">Electronics KTN – Knowledge For Growth</p>	<p>So far we have looked at systems that cope most easily with thermal resistances, whether conduction or convection, but haven’t explicitly looked at airflow. Yet we need this information, for example for sizing fans using something better than trial-and-error. Flow Network Modelling is such a way of estimating airflow within structures, and particularly looking at the distribution of cooling air.</p> <p>Computational Fluid Dynamics is an approach that many people will be familiar with in other contexts. The system of interest is analysed into small cells, the partial differential equations relating to heat, mass flow and similar are derived, and solutions for these calculated iteratively, working in 3-dimensions. It’s this approach that we will be focusing on as an “all-purpose” way of getting information about the flow of air and heat.</p>
 <p>Hand calculation</p> <ul style="list-style-type: none"> ▪ Can be performed to provide a quick estimation of the thermal performance of the product ▪ Critical thermal parameters must be established <ul style="list-style-type: none"> ▪ total dissipated power ▪ physical design of the device ▪ boundary conditions applied ▪ Thermal Resistance Networks representing the different heat transfer paths within the system can be modelled <p style="text-align: right; font-size: small;">Electronics KTN – Knowledge For Growth</p>	<p>Looking first at hand calculation, which is still useful to provide a quick estimation of the thermal performance of a product. As with our earlier modelling, we can build a simple two-resistor model of the heat-generating component, take into account the power dissipation, the physical design, and the boundary conditions and use Thermal Resistance Network models to evaluate the different heat transfer paths.</p>

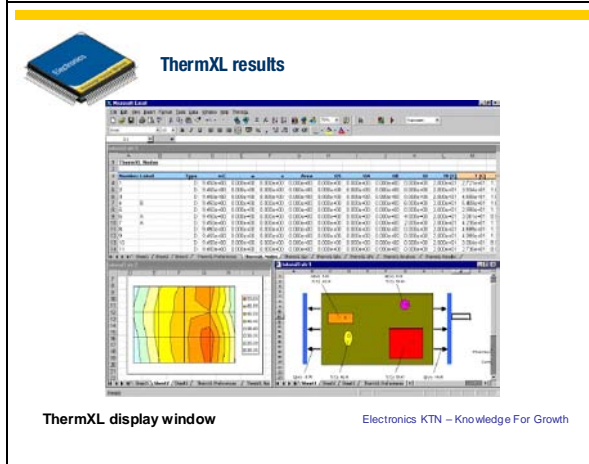
Turning the models into useful form

 <p>Using spreadsheets</p> <ul style="list-style-type: none"> TRNs can become very complex This can result in an extensive number of equations to be solved simultaneously <ul style="list-style-type: none"> often the only way to do this is to use an iterative approach Spreadsheets can handle advanced mathematical and engineering functions and provide graphics capabilities for displaying results <p style="text-align: right; font-size: small;">Electronics KTN – Knowledge For Growth</p>	<p>Particularly with multiple components, Thermal Resistance Networks can become very complex, which is where spreadsheets come into their own. However, TRNs can give large number of equations that need to be solved simultaneously, and often the only way to do this is to use the spreadsheet to carry out these calculations in an iterative fashion. The advantage of the spreadsheet is that many advanced mathematical and engineering functions are built in, and most engineers can use its graph capability to display results in a meaningful fashion.</p>												
 <p>Using spreadsheets</p> <p style="text-align: center;">Compact 2R Models</p>  <table border="1" style="margin-left: auto; margin-right: auto;"> <thead> <tr> <th>Component</th> <th>T_j</th> </tr> </thead> <tbody> <tr> <td>POFP1.5W</td> <td>117</td> </tr> <tr> <td>PLCC</td> <td>131</td> </tr> <tr> <td>TOFP</td> <td>130</td> </tr> <tr> <td>PBGA</td> <td>130</td> </tr> <tr> <td>POFP1.25W</td> <td>109</td> </tr> </tbody> </table> <p style="text-align: center;">T_j calculation from 2R model using Excel</p> <p style="text-align: right; font-size: small;">Electronics KTN – Knowledge For Growth</p>	Component	T _j	POFP1.5W	117	PLCC	131	TOFP	130	PBGA	130	POFP1.25W	109	<p>This example is from a student's submission on one of our case studies and shows the display of junction temperature for a range of different components, based on a two-resistor model. In this case each of these devices was modelled based on quoted values of thermal resistance from junction to case and from junction to board, calculating the junction temperatures from the power dissipations. The aim was to see whether any of the components was likely to exceed its maximum specification temperature, and this simple approach is quite useful for that type of work.</p>
Component	T _j												
POFP1.5W	117												
PLCC	131												
TOFP	130												
PBGA	130												
POFP1.25W	109												
 <p>Add-in package for Microsoft Excel</p> <ul style="list-style-type: none"> The one illustrated is ThermXL™ Uses a 'lumped parameter' modelling technique 1, 2 or 3D system is represented as a series of discrete points (nodes) connected by thermal conductors Can be used for assemblies and not just components <p style="text-align: right; font-size: small;">Electronics KTN – Knowledge For Growth</p>	<p>The next step up is to supplement the computing power of the spreadsheet with an add-in that handles the thermal modelling. The one that we've illustrated is ThermXL. As with DELPHI modelling, it uses a "lumped parameter" technique, where the system is represented by thermal conductors connecting a series of discrete points or nodes at key areas within the structure.</p> <p>The approach can be applied to assemblies and not just components. For example, the board itself can be modelled to take into account conduction through the copper and the materials of the board.</p>												
 <p>Add-in package for Microsoft Excel</p> <ul style="list-style-type: none"> The one illustrated is ThermXL™ Uses a 'lumped parameter' modelling technique 1, 2 or 3D system is represented as a series of discrete points (nodes) connected by thermal conductors Can be used for assemblies and not just components Supports heat transfer by conduction, convection and radiation Limited (by Excel) to 256 nodes <p style="text-align: right; font-size: small;">Electronics KTN – Knowledge For Growth</p>	<p>Thermal resistances can be associated with conduction, convection, and even radiation, although it is normal to disregard radiation, because it is generally insignificant. The approach is only limited in its scope by the constraints of the spreadsheet – with Excel the limitation is 256 nodes.</p>												

Turning the models into useful form



Looking at how the ThermXL model is built up for a board assembly, we first model the board in terms of cells, with a node at the centre of each cell, and adjacent cells joined by conductors that represent the conductance of the board. Devices are then modelled, typically with a two-resistor model, which is joined both to the PCB node and a common “ambient” node.



This shows an example of ThermXL results, with the package operating within Excel, different windows showing both the physical arrangement and heat distribution.

In one of our case studies we look at the results from the tool, which gave reasonable results, so there is some merit in this approach, at least as a first-pass tool for evaluation, before you drag out the great war engine of the CFD-based package!


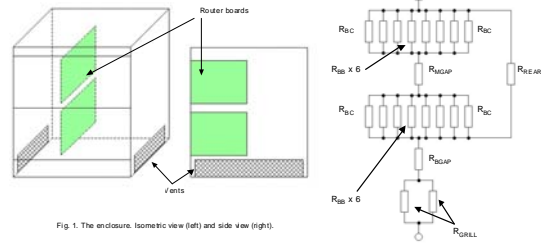


-
- Estimating the airflow**
- Many of the models (especially the simpler ones) concentrate on thermal resistance values
 - Dealing with airflow is much more difficult
 - how much air is needed?
 - what is the resistance to flow of the structure?
 - what fan specification is needed?
 - where should/how can improvements in airflow be made?
- Electronics KTN – Knowledge For Growth

Many of the models, especially the simpler ones, concentrate on thermal resistance values, and these are much easier to apply to conduction, or convection when we apply a nominal value of h . Explicitly dealing with airflow is much more difficult, and we often need to consider for cooling application how much air is needed, and what is the resistance to flow of the structure. This information is needed to scope the specification of the fan, and having information about airflow helps us improve the system.

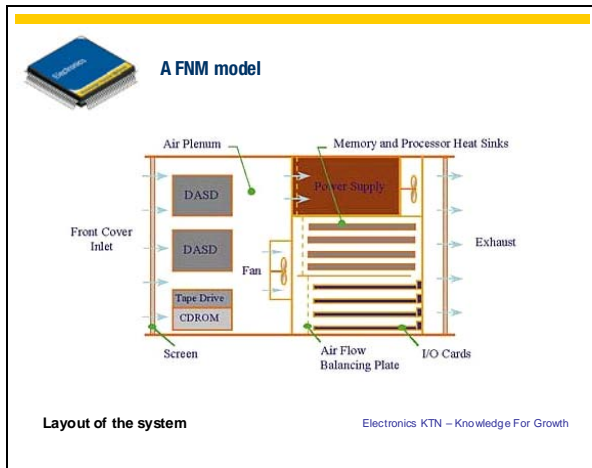
-
- Estimating the airflow**
- Many of the models (especially the simpler ones) concentrate on thermal resistance values
 - Dealing with airflow is much more difficult
 - how much air is needed?
 - what is the resistance to flow of the structure?
 - what fan specification is needed?
 - where should/how can improvements in airflow be made?
 - Analytical techniques
 - start by breaking down the flow resistance into a network
 - estimate flow resistance using basic physics principles
 - can be done manually, or more formally using software
- Electronics KTN – Knowledge For Growth

For this, whilst CFD is an obvious solution, analytical techniques are available that give reasonable estimates, and these start by breaking down the flow resistance into a network, and estimating the flow resistance of each element using basic physics principles. This can be approached manually, or carried out more formally and extensively using software.

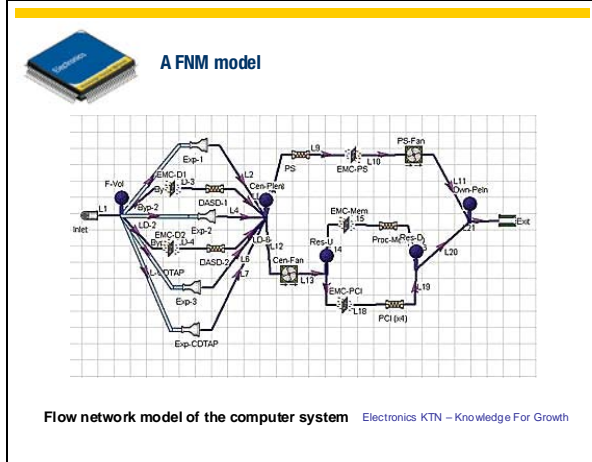
Turning the models into useful form

<div style="display: flex; align-items: center;">  <div> <h3 style="margin: 0;">Hand calculation for airflow modelling</h3> </div> </div> <div style="text-align: center; margin-top: 20px;">  </div> <div style="display: flex; justify-content: space-between; margin-top: 10px;"> <div style="font-size: small;"> <p>Fig. 1. The enclosure. Isometric view (left) and side view (right).</p> <p>Airflow resistance model of telecoms rack</p> </div> <div style="font-size: x-small; color: blue;"> <p>Electronics KTN – Knowledge For Growth</p> <p>Fig. 2. The enclosure resistance network</p> </div> </div>	<p>The illustration here shows the same telecommunication enclosure that you will see in one of the case studies, with two sub-racks each containing seven boards. In this case, we constructed a set of resistances, but in this case they are not thermal resistance, but rather resistance to air-flow.</p> <p>Calculating each of the individual resistances on the basis of its dimensions gave a total air resistance for the complete system. A second piece of information was an estimate of the air-flow needed, derived from information on power dissipation and the acceptable rise in temperature within the equipment. The two parameters of air-flow and resistance could then be compared against the fan characteristics in order to make the correct selection. Whilst this approach makes many assumptions, and the end result is not totally accurate, it gives an initial figure that can be refined.</p>
<div style="display: flex; align-items: center;">  <div> <h3 style="margin: 0;">Flow Network Modelling (FNM)</h3> </div> </div> <ul style="list-style-type: none"> Practical electronic cooling systems can be considered as networks of flow paths through components such as screens, filters, fans, ducts, heat sinks, power supplies The flow network is constructed by graphically representing the paths followed by the flow streams as they pass through different components of the system FNM is a useful tool to quickly estimate the average temperatures of sub-systems and compare the heat transfer efficiency of different designs Can be used in the early phase of product design for a more detailed analysis than is possible by hand calculation/spreadsheets <div style="text-align: right; font-size: x-small; color: blue; margin-top: 10px;"> <p>Electronics KTN – Knowledge For Growth</p> </div>	<p>Flow Network Modelling represents a similar type of tool, looking at the flow of air through a system, and building the flow network graphically in the same way as the manual approach in the last slide. Flow Network Modelling is a useful tool for early stage evaluation, to estimate average temperatures, heat transfer efficiency and similar, and to select appropriate designs. It's a stage more detailed than the hand calculation or spreadsheet idea, but is still an early phase tool, as we will see from the example.</p>
<div style="display: flex; align-items: center;">  <div> <h3 style="margin: 0;">Flow Network Modelling (FNM)</h3> </div> </div> <ul style="list-style-type: none"> FNM techniques are easy to master as long as the 2D airflow paths can be clearly defined The characteristics of these components in terms of their flow and thermal resistances can be obtained from handbooks, vendor specs, or in-house testing The technique does not give detailed results for predicting component temperatures The FNM approach can provide a fast and accurate prediction of the flow distribution and the resulting thermal performance of the system <div style="text-align: right; font-size: x-small; color: blue; margin-top: 10px;"> <p>Electronics KTN – Knowledge For Growth</p> </div>	<p>The concept is quite easy to master, in the same way as thermal resistance, and the same arguments apply about uncertainty as to their value. Information about designs can be gleaned from published information, but may also need measurement. The technique doesn't give a detailed temperature result, but it does provide a quick way of looking at the flow distribution and the resulting thermal performance of the system.</p>

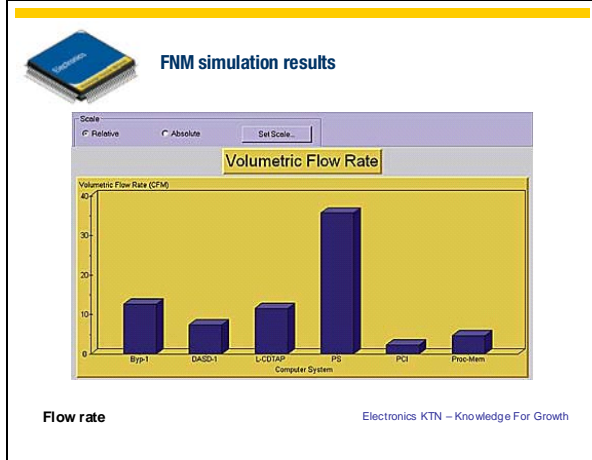
Turning the models into useful form



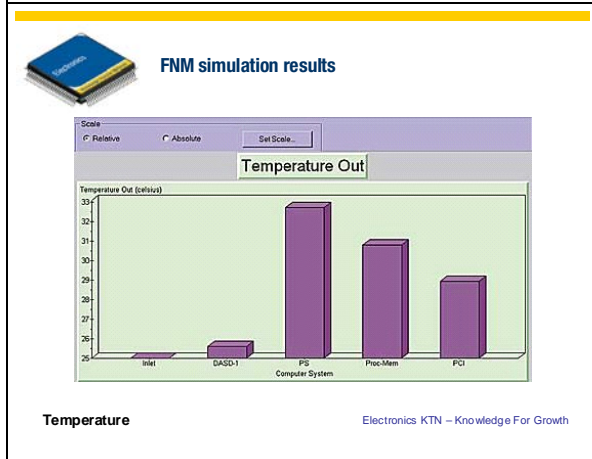
This is a very simple and rather old-fashioned example of FNM applied to a very old style of computer. We look first at the system ...



... and then at the Flow Network Model which this has generated. Each of the key elements is shown.


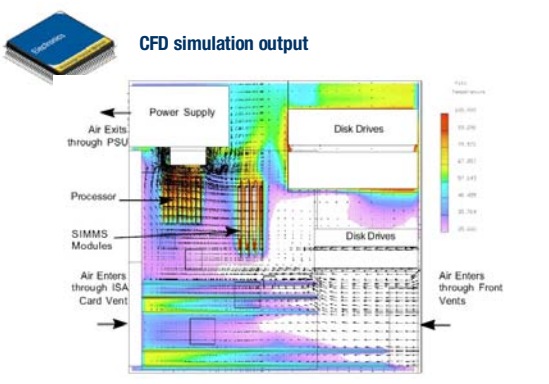



From this, we can look at the flow rate in the different areas of the product, and from this ...




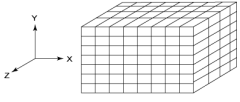


... calculate the temperature rise. Given some software support, this is an approach that will enable the designer to identify at an early stage problems that need to be addressed and looked at in more detail, but it's only really a first pass at system block level, leading on to further exploration using another tool.


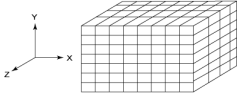

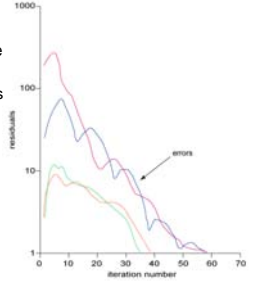

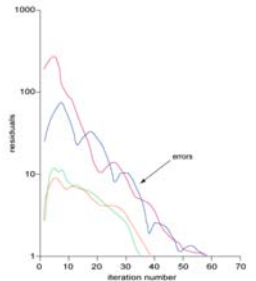

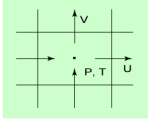
Turning the models into useful form

 <h3>Computational Fluid Dynamics (CFD)</h3> <ul style="list-style-type: none">CFD is a sophisticated <i>simulation</i> techniqueRequires the creation of a computational model that represents a system or device that is to be analysedA CAD system applies fluid-flow and thermal physics to this virtual prototypeThe software outputs a prediction of the fluid dynamics and temperature distribution <p>Electronics KTN – Knowledge For Growth</p>	<p>All the techniques we've looked at so far make significant simplifications in order to be able to create a model that can be computed in a straightforward fashion. By contrast, Computational Fluid Dynamics is a sophisticated simulation technique aimed at producing highly accurate results, which are the result of building a computer model of the system or device to be analysed that represents it in some detail. Fluid flow and thermal physics are applied to what is in effect a virtual prototype, and the resulting partial differential equations are solved by numerical methods.</p> <p>The resulting information, prediction of the fluid flow and temperature distribution, is then presented graphically to the designer. So the CFD tool embeds the three stages of building the model, computing a best fit for its characteristics, and presenting the information in a meaningful form.</p>
 <h3>CFD simulation output</h3> <p>Power Supply, Disk Drives, Processor, SIMMS Modules, Air Exits through PSU, Air Enters through ISA Card Vent, Air Enters through Front Vents</p> <p>Electronics KTN – Knowledge For Growth</p>	<p>This is a typical simulation output from a CFD tool, showing both air-flow and temperature, with an overlay indicating the elements within the system, in this case a PC. Both flow constrictions for the air and hot spots are readily visible.</p>
 <h3>What a CFD system does . . .</h3> <ul style="list-style-type: none">Simulates heat transfer (by conduction, convection and radiation) using the heat conservation equationsSimulates air flow (natural and forced convection) using the Navier-Stokes equations<ul style="list-style-type: none">a set of partial differential equations that describe the motion of viscous fluids such as liquids and gases <p>Electronics KTN – Knowledge For Growth</p>	<p>Internally, what a CFD system does is to simulate both heat transfer and air-flow. Heat transfer, whether by conduction, convection or radiation, is simulated using the heat conservation equations, which express the self-evident truth that heat flow in and out of a volume need to be in balance, or will result in the volume heating or cooling.</p> <p>Air-flow, whether by natural or forced convection, and whether the flow is laminar or turbulent, can be simulated using the Navier-Stokes equations. This is a set of partial differential equations that describe the motion of any viscous fluid, so the principle can apply both to liquid and gases, even though the parameters are considerably different.</p>


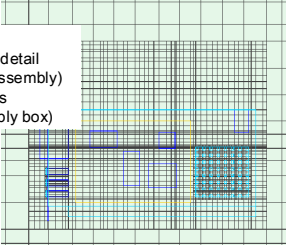

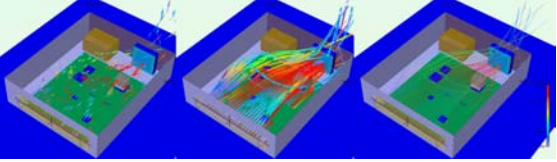


Turning the models into useful form

 <p>Navier-Stokes equations</p> <ul style="list-style-type: none"> ▪ The Navier–Stokes equations <ul style="list-style-type: none"> ▪ are derived by applying the three conservation laws of mass, momentum and energy (Newton's Second Law) to an arbitrary 'control volume' ▪ establish relationships between the <i>rates of change</i> of the variables of interest (such as velocity and pressure) <p style="text-align: right; font-size: small;">Electronics KTN – Knowledge For Growth</p>	<p>The Navier-Stokes equations are derived by applying the three conservation laws of mass, momentum and energy to an arbitrary “control volume”. What they do is to establish relationships between rates of change of the variables of interest, such as velocity and pressure rather than relationships between the variables themselves. For example the Navier-Stokes equations for an ideal fluid will relate acceleration to the pressure gradient.</p>
 <p>Navier-Stokes equations</p> <ul style="list-style-type: none"> ▪ The Navier–Stokes equations <ul style="list-style-type: none"> ▪ are derived by applying the three conservation laws of mass, momentum and energy (Newton's Second Law) to an arbitrary 'control volume' ▪ establish relationships between the <i>rates of change</i> of the variables of interest (such as velocity and pressure) ▪ The Navier-Stokes and heat conservation equations <ul style="list-style-type: none"> ▪ are differential equations that are essentially not solvable for practical cases, <i>but</i> ▪ may be applied to small volumes and 'discretised' to produce algebraic equations that are much easier to solve ▪ values of temperature, pressure and velocity within the control volume can then be calculated <p style="text-align: right; font-size: small;">Electronics KTN – Knowledge For Growth</p>	<p>Both the Navier-Stokes and heat conservation equations are differential equations that are essentially not solvable for practical cases. What we need to do is take those differential equations and discretise them to produce algebraic equations that are much easier to solve, and that is done by splitting the region of interest into many small volumes. Values of temperature, pressure and velocity within the control volume can then be calculated.</p>
 <p>What a CFD system does . . .</p> <ul style="list-style-type: none"> ▪ The space being represented in the model (the 'solution domain') is split up into many small volumes over each of which the conservation equations are expressed in algebraic form <p style="text-align: right; font-size: small;">Electronics KTN – Knowledge For Growth</p>	<p>So the first stage is to split up the model into very many small volumes about which we will make simplifying assumptions, for example that the heat source will be uniform over the entire cell volume, the velocity will be uniform over a cell face and so on. This allows us to express the conservation equations in algebraic form.</p>
 <p>What a CFD system does . . .</p> <ul style="list-style-type: none"> ▪ The space being represented in the model (the 'solution domain') is split up into many small volumes over each of which the conservation equations are expressed in algebraic form ▪ These volumes (referred to as 'control cells', 'grid cells', or just 'cells') need to be <ul style="list-style-type: none"> ▪ non-overlapping ▪ contiguous ▪ finite <div style="text-align: center;">  </div> <p style="text-align: right; font-size: small;">Electronics KTN – Knowledge For Growth</p>	<p>In order to capture the variation of these parameters over the whole region of interest we have to split the system into many small volumes, which is why we need a matrix with many, many cells in it. These volumes are referred to variously as “control cells”, “grid cells” or just “cells”. Cells need to be non-overlapping, they need to be contiguous (that is they need to touch each other, so the whole volume is filled and there is no discontinuity), and they need to be finite.</p>





Turning the models into useful form

<div style="display: flex; align-items: center;">  <div> <h3>What a CFD system does . . .</h3> <ul style="list-style-type: none"> The space being represented in the model (the 'solution domain') is split up into many small volumes over each of which the conservation equations are expressed in algebraic form These volumes (referred to as 'control cells', 'grid cells', or just 'cells') need to be <ul style="list-style-type: none"> non-overlapping contiguous finite A Cartesian 'mesh' is a common choice, but the mesh does not need to be regular </div> </div> <div style="text-align: center; margin-top: 10px;">  <p style="font-size: small;">Electronics KTN – Knowledge For Growth</p> </div>	<p>There are many ways of splitting up the volume, of which a “Cartesian mesh” is a common choice, but there are several other ways of doing it. The key point is that the mesh does not need to be regular, but can be finer in some areas than others.</p>
<div style="display: flex; align-items: center;">  <div> <h3>Solving the equations</h3> <ul style="list-style-type: none"> The conservation equations are both non-linear and 'coupled': <ul style="list-style-type: none"> the value of a variable depends on surrounding values of that variable and also on other variables Equations need to be solved in an iterative manner, until the errors in the conservation equations are at an acceptable level </div> </div> <div style="text-align: center; margin-top: 10px;">  <p style="font-size: small;">Error convergence with multiple iterations Electronics KTN – Knowledge For Growth</p> </div>	<p>The conservation equations are non linear, but they are also “coupled”, that is the value of a variable depends on the values of that variable in surrounding cells, and also on other variables within the cell, which makes the situation extremely complex.</p> <p>In consequence, equations need to be solved iteratively, that is by estimating solutions, applying them within the equations, re-estimating the solutions, and repeating the process until solutions settle down. This settling-down process is referred to as convergence, and can be seen graphically by monitoring the errors in the conservation equation. We proceed with the iterative solution process until the errors in the conservation equations reach an acceptable level.</p>
<div style="display: flex; align-items: center;">  <div> <h3>Solving the equations</h3> <ul style="list-style-type: none"> The finer the grid used, the higher the cell count and The more closely do the algebraic equations approximate to the original differential equations but The greater is the computer overhead to calculate </div> </div> <div style="text-align: center; margin-top: 10px;">  <p style="font-size: small;">Error convergence with multiple iterations Electronics KTN – Knowledge For Growth</p> </div>	<p>Our aim is convergence, with the residual errors as low as possible, but we also want as accurate a solution as possible, so there is a temptation to use a fine grid, with small cells.</p> <p>However, the finer the grid used, the higher is the cell count. This has the advantage that the algebraic equations for the cell approximate more closely to the original differential equations, but the considerable downside that the more cells, the greater is the computer overhead to calculate the results. Setting too fine a grid is probably the single greatest contributor to simulations failing to converge.</p>
<div style="display: flex; align-items: center;">  <div> <h3>The grid cell</h3> <ul style="list-style-type: none"> The calculated values for field variables such as temperature and pressure are located at the centres of the grid cells The values of the velocity components are located at the faces of the grid cells A key concept within grid configuration is the aspect ratio, which is the relationship between length (L1) and width (L2) of the grid cells. As a general rule, this ratio should be as close to unity as possible: <ul style="list-style-type: none"> L1/L2 = 1 is ideal L1/L2 < 20 is acceptable L1/L2 > 200 is likely to cause solution convergence issues </div> </div> <div style="text-align: center; margin-top: 10px;">  <p style="font-size: small;">Electronics KTN – Knowledge For Growth</p> </div>	<p>Looking in more detail at each grid cell, the calculated values for field variables, such as temperature and pressure, are located at the centres of the grid cells, whereas the values of the velocity components are located at their faces.</p> <p>Very rarely do grids take the regular cubic shape that we have indicated earlier, and it is normal for cells to be elongated or flattened. So a key concept when configuring a grid is the relationship between length and width of the cells, referred to as the aspect ratio. As a general rule, this ratio should be as close to unity as possible, but values of under 20 are acceptable, whereas very large values are likely to cause problems in getting the solution to converge.</p>




Turning the models into useful form

 <h3>A typical grid mesh</h3> <ul style="list-style-type: none">▪ Illustrates the non-uniform distribution of cells across a sub-assembly <p>The grid is</p> <ul style="list-style-type: none">▪ densest across regions of high detail (for example, a small internal assembly)▪ coarsest across low detail areas (that is, outside the sub-assembly box)  <p>Electronics KTN – Knowledge For Growth</p>	<p>A typical grid meshing for a product is illustrated here. Notice the non-uniform distribution of cells across the assembly. The grid is densest across regions of high detail, for example the small internal assembly, and coarsest in areas where little is happening, such as outside the sub-assembly box.</p>
 <h3>What a CFD system does . . .</h3> <ul style="list-style-type: none">▪ Information prepared for presentation once the solutions have converged sufficiently▪ Graphical information displayed shows both airflow and temperature distribution  <p>Electronics KTN – Knowledge For Growth</p>	<p>When the grid has been produced, and the equations solved, the CFD system now prepares information for presentation. But the process has to wait until the solutions have converged sufficiently, which may take some time!</p> <p>Typically information is displayed graphically, and shows both air-flow and temperature distribution. For air-flow, one usually has options of showing the flow direction, or just the flow volume. Both temperature and airflow are normally colour-coded, the rainbow showing highest temperatures or greatest air movement at the red end.</p>
 <h3>Using CFD tools</h3> <ul style="list-style-type: none">▪ Calculation time can be extensive with complex/high detail designs <p>Electronics KTN – Knowledge For Growth</p>	<p>Using CFD tools can be challenging in terms of computation time, with reports of simulations that have run all weekend! This is where careful preparation is important, in particular in constructing the grid. There needs to be an overriding awareness that the task will be computationally demanding, that results can take some considerable time to achieve, and that the size of the problem needs to be managed appropriately.</p>
 <h3>Using CFD tools</h3> <ul style="list-style-type: none">▪ Calculation time can be extensive with complex/high detail designs▪ Needs careful attention to gridding to reduce computation time and get reliable results<ul style="list-style-type: none">▪ grid too coarse = poor accuracy of prediction▪ grid too fine = massive computational overhead▪ grid too fine = may fail to converge <p>Electronics KTN – Knowledge For Growth</p>	<p>One of the crucial aspects in using CFD tools is the computational grid, bearing in mind that the differential equations have to be solved for each grid cell, and not independently, but as a complete three-dimensional array. If the grid applied to the overall system is too coarse, the accuracy of prediction will be poor; if it is too fine, you are faced with a massive computational overhead and there is the worse problem that it may fail to converge.</p> <p>Correct choice of grid is crucial, but fortunately most tools have inbuilt facilities to ease gridding, and to localise the grid over key areas such as critical components, but with a coarser grid elsewhere.</p>

Turning the models into useful form

 <p>Using CFD tools</p> <ul style="list-style-type: none">▪ Calculation time can be extensive with complex/high detail designs▪ Needs careful attention to gridding to reduce computation time and get reliable results<ul style="list-style-type: none">▪ grid too coarse = poor accuracy of prediction▪ grid too fine = massive computational overhead▪ grid too fine = may fail to converge▪ Can also be helped by building into the model simplified (and validated) models of key elements <p style="text-align: right;"><small>Electronics KTN – Knowledge For Growth</small></p>	<p>Both computing time and accuracy of convergence can also be assisted by building into the model simplified models of key elements, which is where things like DELPHI models come in. Of course, they need to be validated, as wrong models lead to wrong solutions.</p>
 <p>Simulation software</p> <ul style="list-style-type: none">▪ Varies significantly in its functionality although the sequence of modelling a project remains the same<ul style="list-style-type: none">▪ importing mechanical, electrical and thermal data▪ gridding▪ solving▪ analysing the results <p style="text-align: right;"><small>Electronics KTN – Knowledge For Growth</small></p>	<p>Simulation software varies very significantly in its functionality, although the sequence of modelling a project remains the same. The starting point is importing information on the mechanical aspects of the assembly, on the dissipation of the components and on their thermal characteristics. A grid is then applied and optimised, and solutions to the partial differential equations produced iteratively. Once the solutions have converged, the results are presented.</p>
 <p>Simulation software</p> <ul style="list-style-type: none">▪ Varies significantly in its functionality although the sequence of modelling a project remains the same<ul style="list-style-type: none">▪ importing mechanical, electrical and thermal data▪ gridding▪ solving▪ analysing the results▪ Software solutions are available for<ul style="list-style-type: none">▪ importing mechanical/electrical CAD-data directly into the CFD software<ul style="list-style-type: none">▪ helps avoid divergence between design and simulation▪ assisting the matching between grid and application <p style="text-align: right;"><small>Electronics KTN – Knowledge For Growth</small></p>	<p>Either built into the tool, or available as separate software solutions, are means of carrying out two key functions. The first of these is importing design data directly into the simulation software, so that the simulation model is as accurate as possible a reflection of the current design. It is not just a question of saving time rebuilding the model mechanically, but more a question of making sure that the simulation is working from the same database as the design, which can be very important when the design is subject to frequent change. A second area where software assistance is available is in the matching of the grid to the application.</p>
 <p>Turning the models into useful form</p> <p style="text-align: center;">recommend getting advice and trying a range of tools</p> <ul style="list-style-type: none">▪ There are many different thermal analysis approaches, and a wide choice of simulation tools<ul style="list-style-type: none">▪ software cost can range from £100s to £1,000s▪ trade-off between cost, functionality and ease of use <p style="text-align: right;"><small>Electronics KTN – Knowledge For Growth</small></p>	<p>There are many different ways of tackling thermal analysis and a wide range of simulation tools. Software can cost hundreds of pounds or thousands of pounds, depending on its functionality, and there are, as you would expect with any software, a number of trade offs to be considered between cost, functionality, and ease of use. Which tool is best for you will depend on both your previous experience – it's easier to get good results from a tool that you know – and the application, as well of course as the budget available. So we always recommend seeking advice, and trying a range of tools for your specific application.</p>

Turning the models into useful form

 <p>Turning the models into useful form</p> <p>recommend getting advice and trying a range of tools</p> <ul style="list-style-type: none">▪ There are many different thermal analysis approaches, and a wide choice of simulation tools<ul style="list-style-type: none">▪ software cost can range from £100s to £1,000s▪ trade-off between cost, functionality and ease of use▪ Tools can be used progressively throughout design cycle<ul style="list-style-type: none">▪ for challenging thermal situations, using tools even as early as the concept phase is recommended as highly effective▪ communication between players is vital for success <p>Electronics KTN – Knowledge For Growth</p>	<p>What is important is to recognise that tools can be used throughout the design cycle, and are best applied early to detect potential problems, rather than to deal retrospectively with issues. Given a challenging thermal situation, we would recommend using tools right back in the concept phase, given that 80% of the eventual product costs are determined early on, and thermal management can be a significant cost area, particularly if applied too late.</p> <p>And a final reminder is that it is important for everyone involved in the design process to be committed to using the tools, and sharing information, so that their use is as effective as possible.</p>
 <p>Turning the models into useful form</p> <ul style="list-style-type: none">▪ There are many different thermal analysis approaches, and a wide choice of simulation tools▪ Tools can be used progressively throughout design cycle▪ With increasing thermal challenges, software tools are becoming more commonly used <p>Electronics KTN – Knowledge For Growth</p>	<p>In summary, there are many different thermal analysis approaches, and a wide range of simulation tools available, and these tools can be used progressively throughout the design cycle. Generally, the earlier you start, the better!</p> <p>As electronic products are getting more challenging thermally, the use of software tools to simulate products is becoming much more common.</p>
 <p>Turning the models into useful form</p> <ul style="list-style-type: none">▪ There are many different thermal analysis approaches, and a wide choice of simulation tools▪ Tools can be used progressively throughout design cycle▪ With increasing thermal challenges, software tools are becoming more commonly used▪ Not a universal panacea or instant solution<ul style="list-style-type: none">▪ no substitute for engineering common sense▪ need to understand the limitations of the techniques▪ garbage in = garbage out!▪ sophisticated software has a considerable learning curve <p>Electronics KTN – Knowledge For Growth</p>	<p>However, these tools should not be regarded as a universal panacea, or an instant solution for thermal problems. There is still no substitute for engineering commonsense! Users need to understand the limitations of the techniques, and particularly to remember that bad data leads to bad simulations and bad conclusions – garbage in = garbage out!</p> <p>Finally, users must remember that sophisticated software has a considerable learning curve, and there is unfortunately no way that just buying the latest XYZ CFD tool will make you an instant thermal guru.</p>