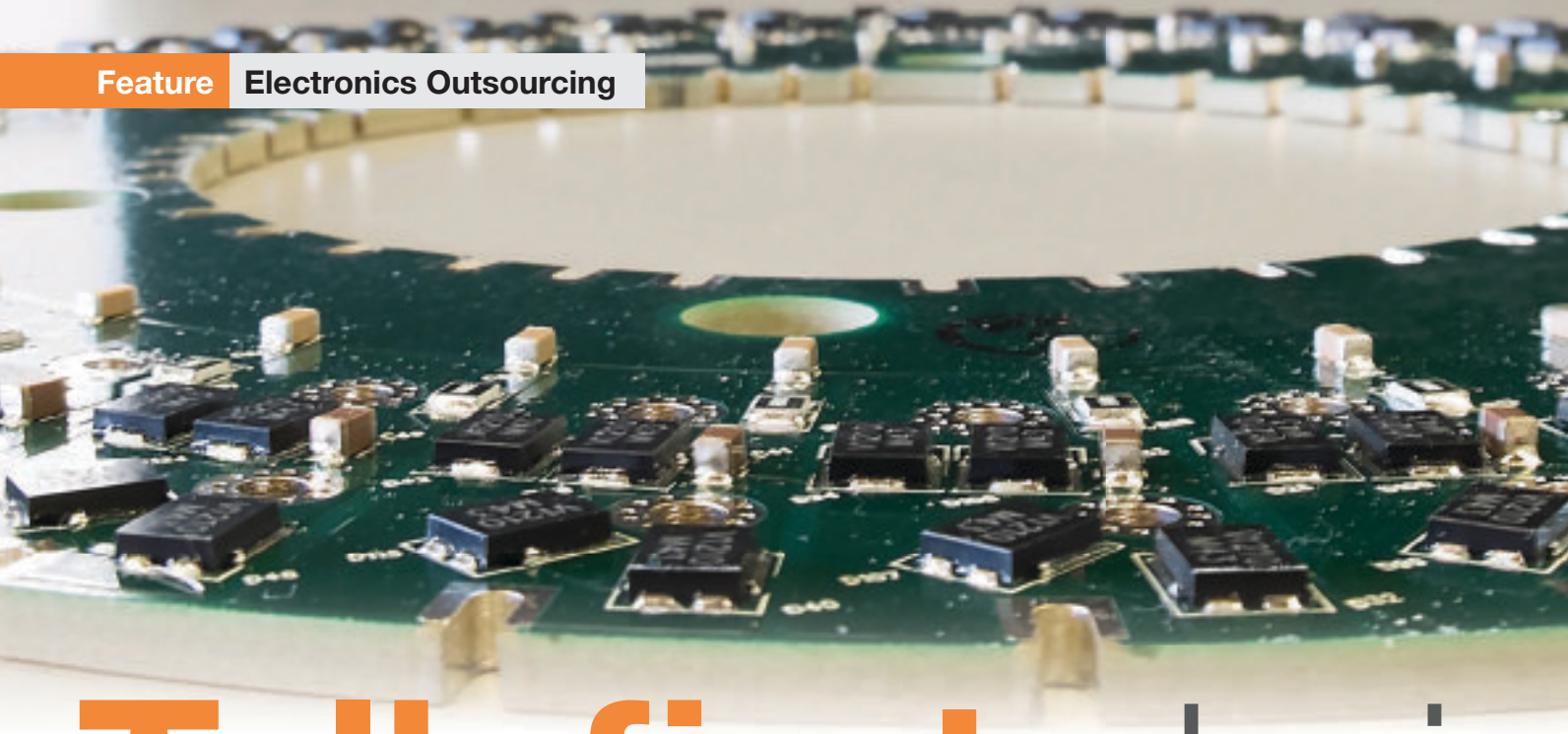

Talk first, desig

If you've developed a new product that has an electronic subassembly, the temptation to quickly design a PCB layout and get it into production is almost irresistible – especially if you have a customer who's demanding delivery tomorrow... or yesterday! But this approach often turns out to be a case of more haste, less speed – and a lot less profit, says ***David Dickin, Business Development Director at Prism Electronics****.*

These days, electronic subsystems are used in an enormous variety of products, and the engineers who design these subsystems are usually talented and resourceful. In many cases, however, their primary interest is functionality, rather than manufacturability. To put it plainly, coming up with the PCB design can be a bit of a bore, and definitely not something they typically want to spend much time on – particularly as the sooner the board is designed, the sooner they can get their hands on a prototype!

This is a shame, because as an experienced contract electronics manufacturer (CEM), Prism Electronics sees the downside of this approach all too often. While in most cases, the boards can be manufactured, and can usually be made to work, questions remain: how long do they take to make? How much do they cost? And how long will they be able to remain in production? All important questions, but all too often, they are not asked until it's too late.

Let's consider some potential pitfalls. A poor board layout with, for example, tiny components next to much larger ones, can potentially complicate placement – possibly even creating a need for costly and time-consuming manual rework. And if the CEM producing the board uses automated optical inspection (AOI) – which is a huge benefit and major cost saver – the AOI system may struggle to see the very small component in the shadow of its much larger neighbour. Manual inspection will then be needed, which again increases manufacturing time and adds to costs.

So why don't we suggest improvements as soon as we see the proposed board layout? The fact is that we do, but it's often too late, especially when regulatory approvals are involved.

For Prism, when we see problems like this, the really disappointing thing is that the issues could almost always have been fixed easily and inexpensively at the board design stage.

So why don't we suggest improvements as soon as we see the proposed board layout? The fact is that we do, but it's often too late, especially when regulatory approvals are involved. If you've just invested tens of thousands of pounds to achieve EMC approval for the piece of equipment that includes the board, for instance, you're far less likely to be receptive to the suggestion of a board redesign that will invalidate those approvals.

Then there are the components themselves. It's up to the designer, of course, to select the best component for the job, but rarely does that come down to just one possibility – often there are multiple components that could be used, and the final choice is sometimes made somewhat arbitrarily, often on the basis of little more than personal preference.

n later!

But that's not always a good criterion! Some of the options may be much easier for a pick-and-place board assembly machine to handle than others, and some may be far easier to source than their functionally equivalent counterparts. It's often good practice to try to avoid single-source components, for example, as supplies could potentially disappear overnight if the one and only manufacturer were to cease production.

There is also obsolescence to consider. If you're introducing a new product, you no doubt hope to continue producing it for some considerable time. Even after that, you're likely to need, or be obliged, to supply spares for years to come.

So designing in a component that's close to the end of its lifecycle, and may shortly be discontinued, is definitely not a good idea. While you may have the opportunity of a 'last-time buy' when production of the component ceases, how many should you buy, and how much money do you want to tie up in stock that you may or may not use?

If you've read this far, you may well be despairing that designing a good PCB layout is a costly and difficult process that's going to put a big strain on your resources, however desirable the outcome may be. Fortunately, this doesn't have to be the case, as there's a convenient, cost-effective and dependable solution to the board design challenge. It comes in two parts.

The first part is to work with a CEM that has wide and demonstrable experience of producing high quality products, as well as an experienced in-house team of engineers and technicians who speak your language and understand your problems.

The second part of the solution – and it's arguably just as important as the first – is to talk first and design later. In other words, talk to your CEM at the earliest possible stage in the development of your new product. The CEM will then use its experience to guide you through the pitfalls of board design, component sourcing and looming obsolescence, resulting in a board that can be manufactured and tested rapidly and economically – not just today, but well into the future.

Of course, even if you're already past the initial concept stage, you should still talk to your CEM, but remember that the later you do so, the more difficult it will be to incorporate its suggestions. Above all, start the dialogue before you spend a lot a money on obtaining design approvals; because if you leave it until afterwards, it may be almost impossible to act on your CEM's suggestions.

It's true that board design can be challenging and there are many factors, obvious and not so obvious, that will affect whether a board is economical to manufacture and test. But there's no need for you to take on the design burden unaided. Choose the right CEM, involve them in your plans from the outset, and you've cracked it – the challenge, that is, not the board!

Talk first and design later. In other words, involve your CEM at the earliest possible stage in the development of your new product.