

# Re: Eagle holes/pad widening

---

*Source:* <http://sci.tech--archive.net/Archive/sci.electronics.basics/2005-06/msg00734.html>

---

- *From:* "Matthias Melcher" <[no-spame-matt@xxxxxxxxxxxxxx](mailto:no-spame-matt@xxxxxxxxxxxxxx)>
  - *Date:* Fri, 24 Jun 2005 14:25:52 +0200
- 

>> I'm a new user of eagle, and I've managed to get through designing my  
>> schematic to designing a PCB.

> You will probably want to make the minimum larger but play with it.

I beleive I should mention that Cadsoft has multiple great newsgroups on their server too. Very helpful folks there including the authors:  
[news.cadsoft.de](http://news.cadsoft.de)

---

• *Follow-Ups:*

- ◆ [\*\*Re: Eagle holes/pad widening\*\*](#)  
    ◇ *From:* MarkMc

• *References:*

- ◆ [\*\*Eagle holes/pad widening\*\*](#)  
    ◇ *From:* MarkMc
- ◆ [\*\*Re: Eagle holes/pad widening\*\*](#)  
    ◇ *From:* James Morrison

- Prev by Date: [\*\*Re: Eagle holes/pad widening\*\*](#)
- Next by Date: [\*\*Re: Eagle holes/pad widening\*\*](#)
- Previous by thread: [\*\*Re: Eagle holes/pad widening\*\*](#)
- Next by thread: [\*\*Re: Eagle holes/pad widening\*\*](#)
- Index(es):
  - ◆ [\*\*Date\*\*](#)
  - ◆ [\*\*Thread\*\*](#)