Tips to Avoid Oscillation & Noise Problems in Circuit Board Layout

Oscillations come from positive feedback from the output of amplifiers to the input. There are some things you can do out of habit as a matter of good physical circuit design that will save you literally hours and hours fighting oscillations. Here are some of them that I know of.

1) Use a circuit board rather than a breadboard.
2) Use power and ground buses that are larger traces or wires. Ground plane can help with this.
3) Connect the power supply to the output end of your circuit, so that the heavy current drawing part of the amplifier or circuit doesn't cause IR drops near the input of the amplifier through the power buss. The smallest signal input should be the farthest from where the power connection connects to the power supply.
4) Use ground plane fill in on your circuit board layout. Always connect it to ground or a power rail. Using flood fill without connecting it to small signal ground is much worse than not using it at all, because the islands form capacitor bridges between the output and the input.
5) Use an RC bypass low pass filters for each stage of your amplifier or circuit, not just bypass capacitors.
6) Use star connections to ground and the power rails (or to the bypass capacitor of each part of your circuit) as much as possible.
7) Avoid ground loops (especially encircling large areas). This includes loops that are partly small signal ground (which can be made using bypass capacitors to complete the loops). Ground plane can help alleviate this problem.
8) Enclose each circuit its own shielded enclosure.
9) Never run output traces near the input, or input traces.
10) Use shielded cable on the inputs and outputs. Make sure that all current paths reduce inductance (make the area enclosed by the current loop as small as possible by running the return wire right by the signal wire).
11) Use surface mount parts. They have less parasitic inductance and capacitance to couple the input signal to the output.
12) Route your signals using strip line techniques so that the signal is more confined to the area where you are routing it. Stripline is preferable to Microstrip. Stripline is ground plane, signal, ground plane. Microstrip is signal, ground plane.
13) Use the shortest possible wires to connect components.
14) Wrap the power supply wires into a braided rope, so that there is no loop area for changing magnetic fields to penetrate.
15) Use low impedance inputs and outputs. High impedance points have little loading effect on coupled signals. (For example, put your thumb on a scope probe, and note how much 60 Hz noise you can see, then put a 50 ohm resistor across the probe and try it.) Provide examples to show why each of these is necessary.
16) Use low Q capacitors for bypass. High Q along with inductance of power traces can cause resonances.

17) Orient air core coils 45 degrees to each other. This gives a near zero coupling. Never orient them coaxially or side by side if you can help it.
18) Use separate analog and digital ground planes. Often you can use two separate regulators to supply each of these. They are connected at one point near where the power enters the board. Do not run signals across from the digital side of the board to the analog side of the board and vice versa, and if it must be done, run them next to the place that connects the digital to the analog ground planes.
19. Make sure each signal has its parallel (as much as possible) return line or plane. This minimizes ground loops.
20. You can twist the pairs to achieve less ground loop area, even on a circuit board by using two layers for this.
21. Use ground-signal-ground on the input and the output of amplifiers to minimize coupling between the input and output.
22. Use differential signals where possible. It improves the signal to noise by 3 db, and make sure differential lines are symmetrically balanced everywhere they go.
23. Read the data sheets of parts you are using carefully. They often have specific tips regarding circuit board layout.

Worth reading references:
http://www.pcb-prototype.net/article/PCB_Design_Layout.html