



US00RE44586E

(19) **United States**
(12) **Reissued Patent**
Hsu et al.

(10) **Patent Number:** **US RE44,586 E**
(45) **Date of Reissued Patent:** **Nov. 12, 2013**

(54) **VIA STRUCTURE FOR IMPROVING SIGNAL INTEGRITY**

7,360,308 B2 * 4/2008 Oggioni et al. 29/852
2002/0017399 A1 * 2/2002 Chang et al. 174/262
2002/0153167 A1 * 10/2002 Miller 174/261
2008/0185180 A1 * 8/2008 Cheng et al. 174/266

(76) Inventors: **Hsiuan-ju Hsu**, Nan-chou (TW);
Richard Walter Ziolkowski, Tucson,
AZ (US)

FOREIGN PATENT DOCUMENTS

TW 95117616 5/2006

* cited by examiner

Primary Examiner — Jeremy Norris

(21) Appl. No.: **13/609,616**

(22) Filed: **Sep. 11, 2012**

Related U.S. Patent Documents

Reissue of:

(64) Patent No.: **8,084,695**
Issued: **Dec. 27, 2011**
Appl. No.: **11/651,338**
Filed: **Jan. 10, 2007**

(51) **Int. Cl.**
H05K 1/11 (2006.01)
H05K 1/14 (2006.01)

(52) **U.S. Cl.**
USPC **174/262**; 361/794

(58) **Field of Classification Search**
USPC 174/262–266; 361/792–795
See application file for complete search history.

(56) **References Cited**

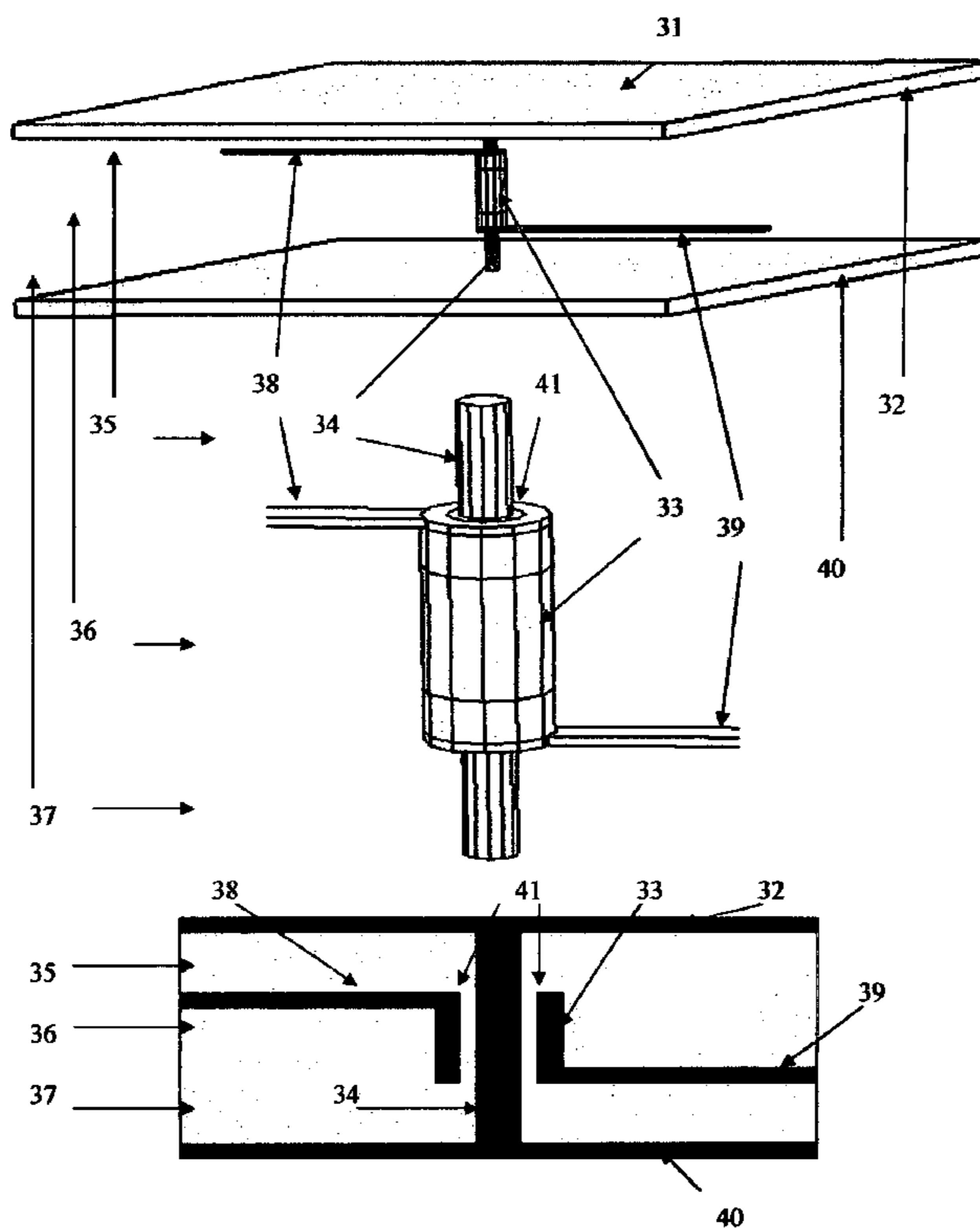
U.S. PATENT DOCUMENTS

5,949,030 A * 9/1999 Fasano et al. 174/262
6,479,764 B1 * 11/2002 Frana et al. 174/262

(57) **ABSTRACT**

The embodiment of the invention is about a novel via structure which can be incorporated into printed circuit boards, integrated circuit packages, and integrated circuits in order to reduce crosstalk, to improve signal integrity and to achieve EM emission compliance. A 4-layer (2 signal layers and 2 power layers or 2 signal layers and 2 ground layers) circuit board assembly was used for demonstrating the effect of the novel via structure. The same concept can be applied to any multi-layer circuit board. Layers that have an electrical property can be added above, under, or within the basic 4-layer circuit board to achieve a multi-layer circuit board. For 2-layer and 3-layer circuit boards, a deformed version of the proposed via structure based upon the same concept will be needed for a coplanar waveguide configuration.

13 Claims, 14 Drawing Sheets



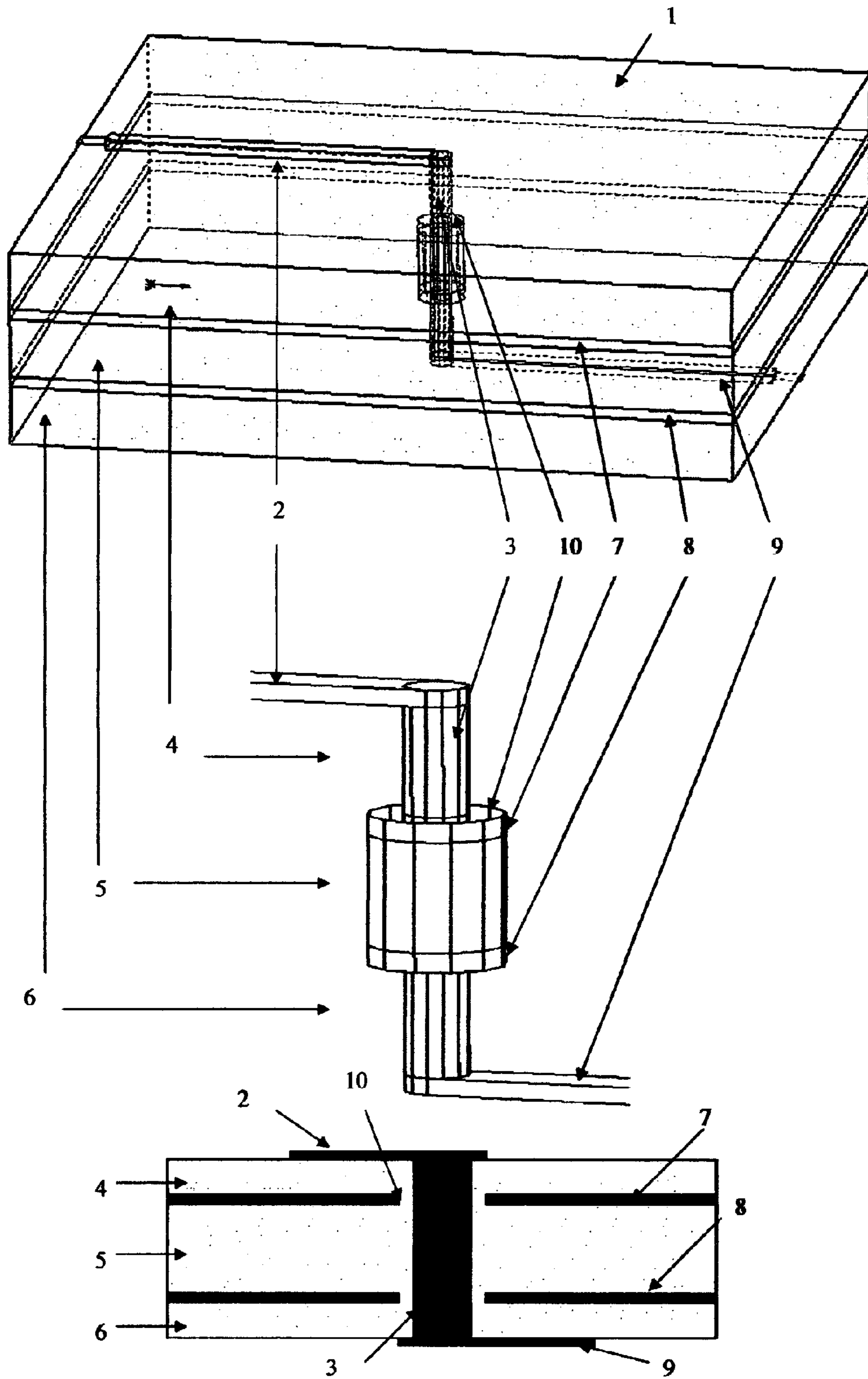
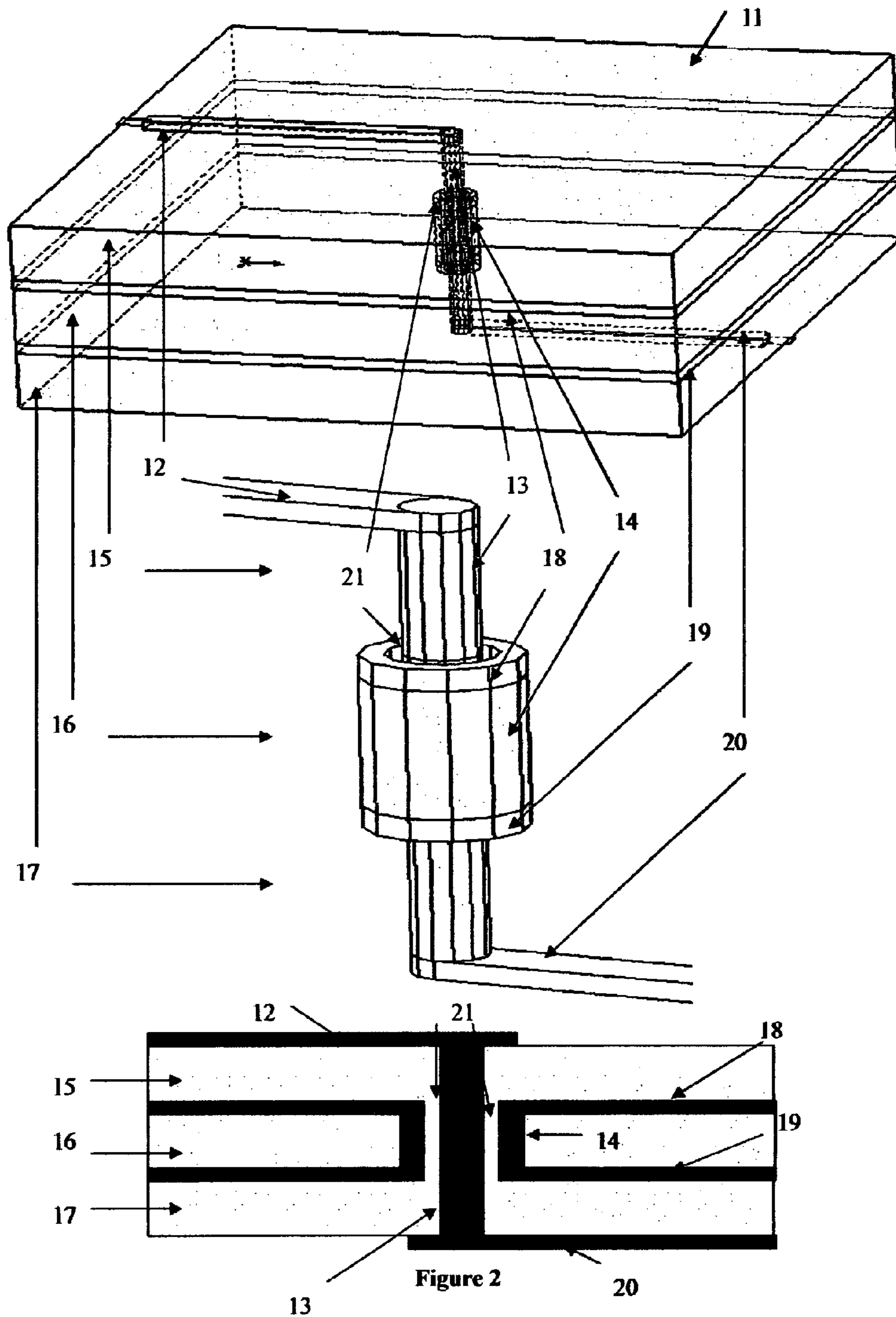


Figure 1 (prior art)



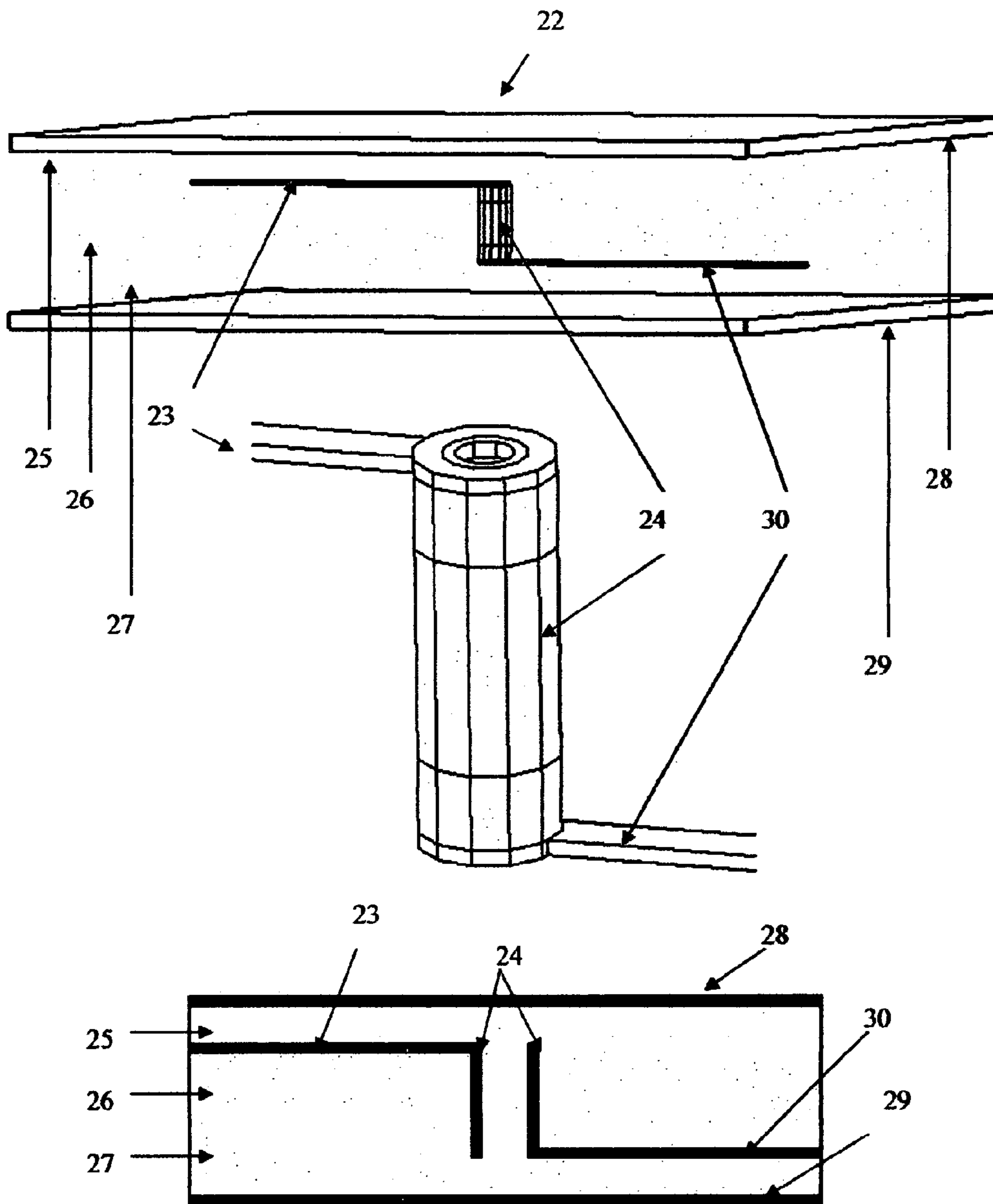


Figure 3 (prior art)

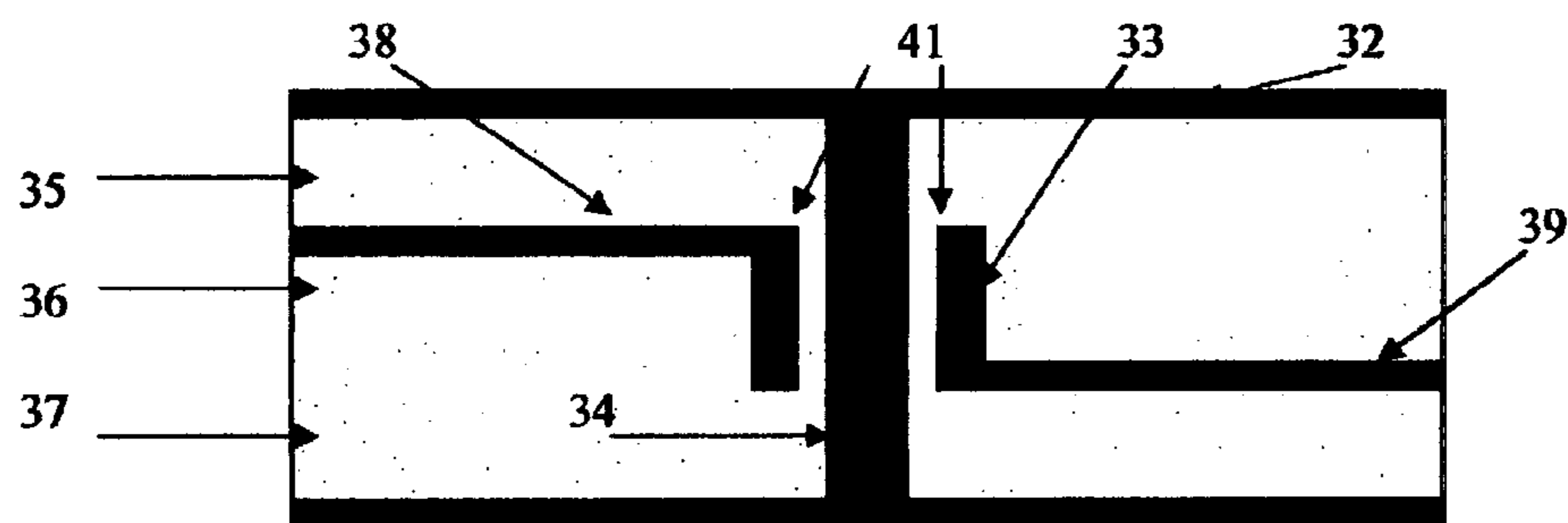
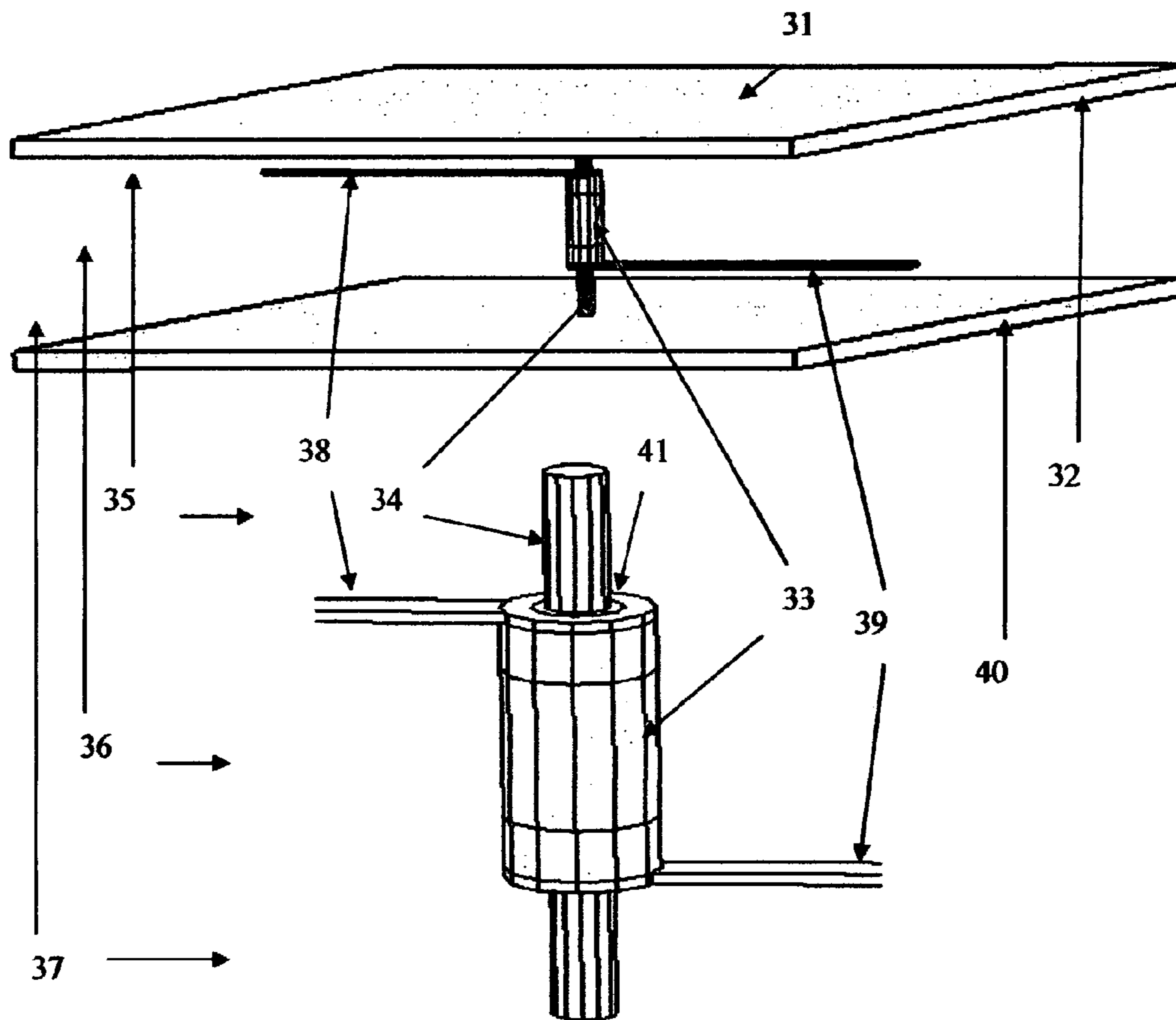


Figure 4

40

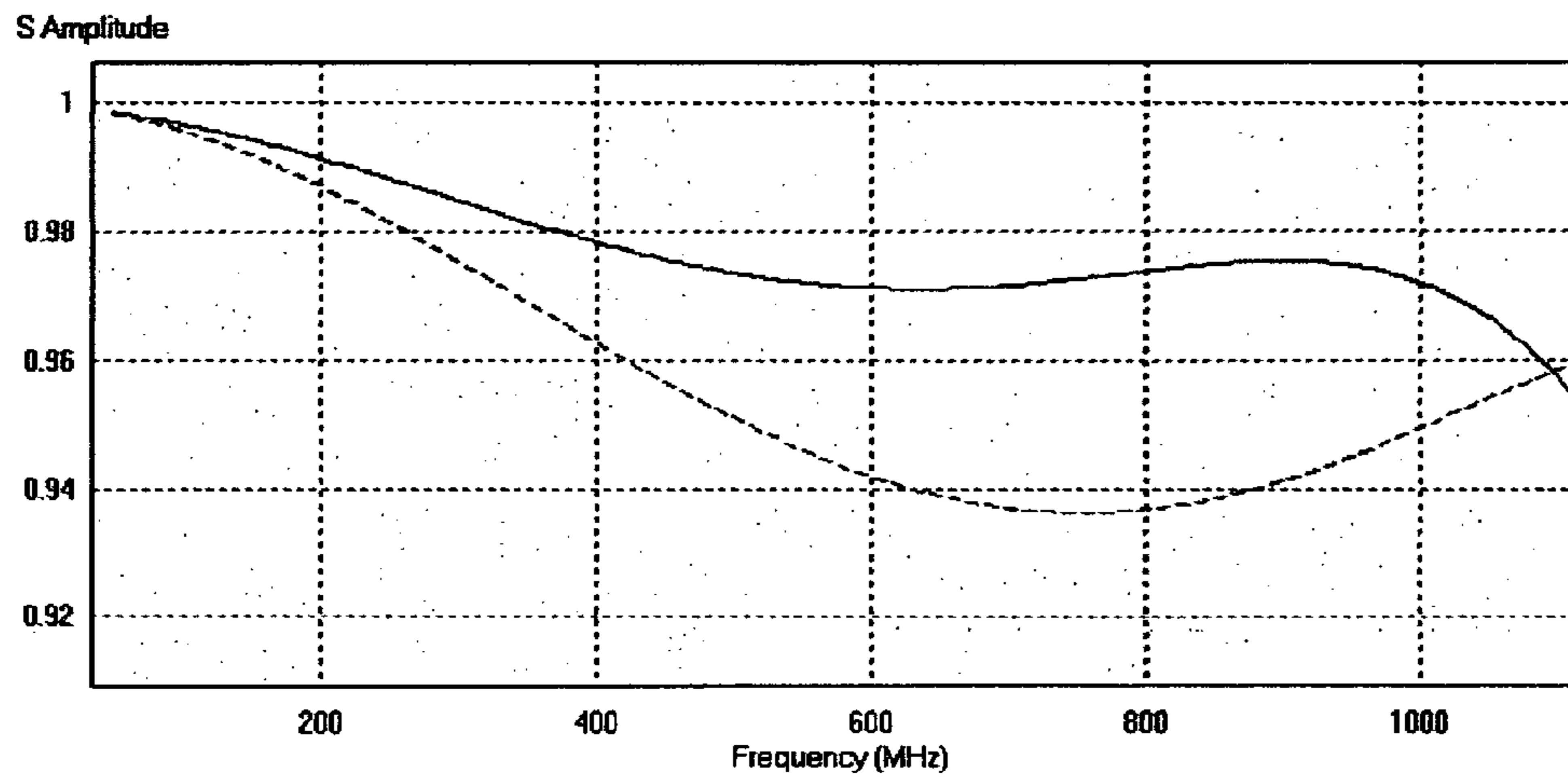


Figure 5

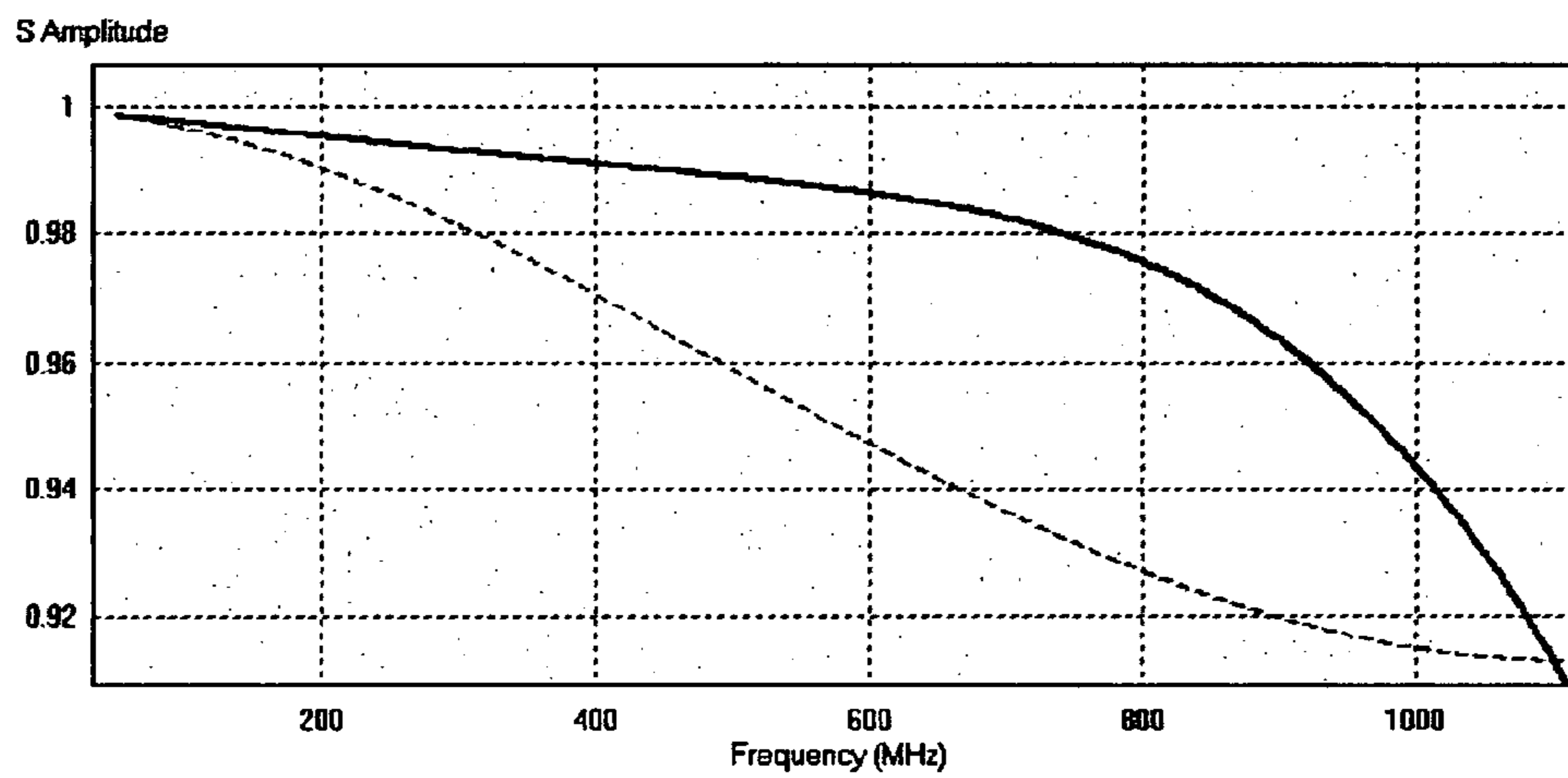


Figure 6

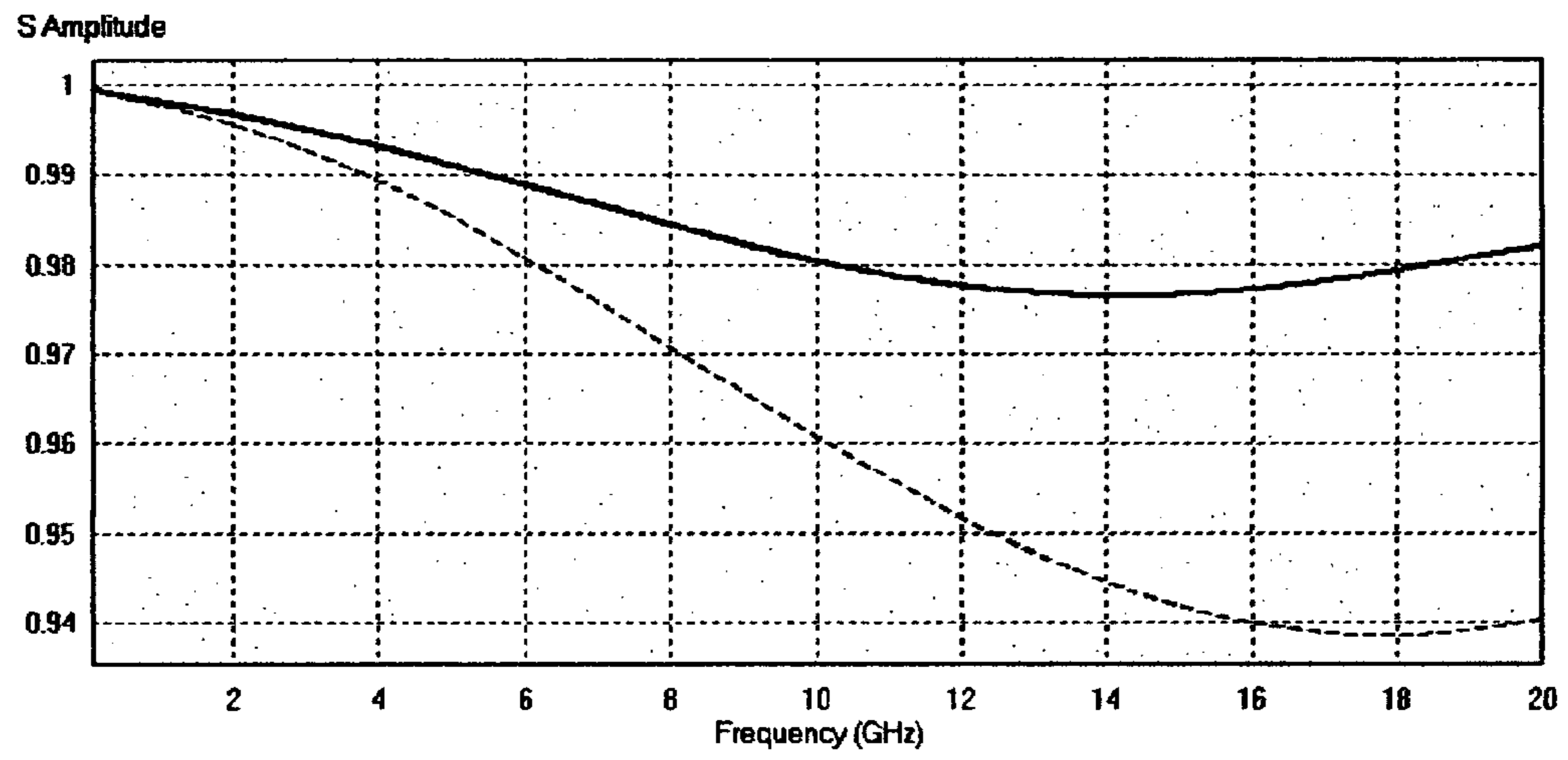


Figure 7

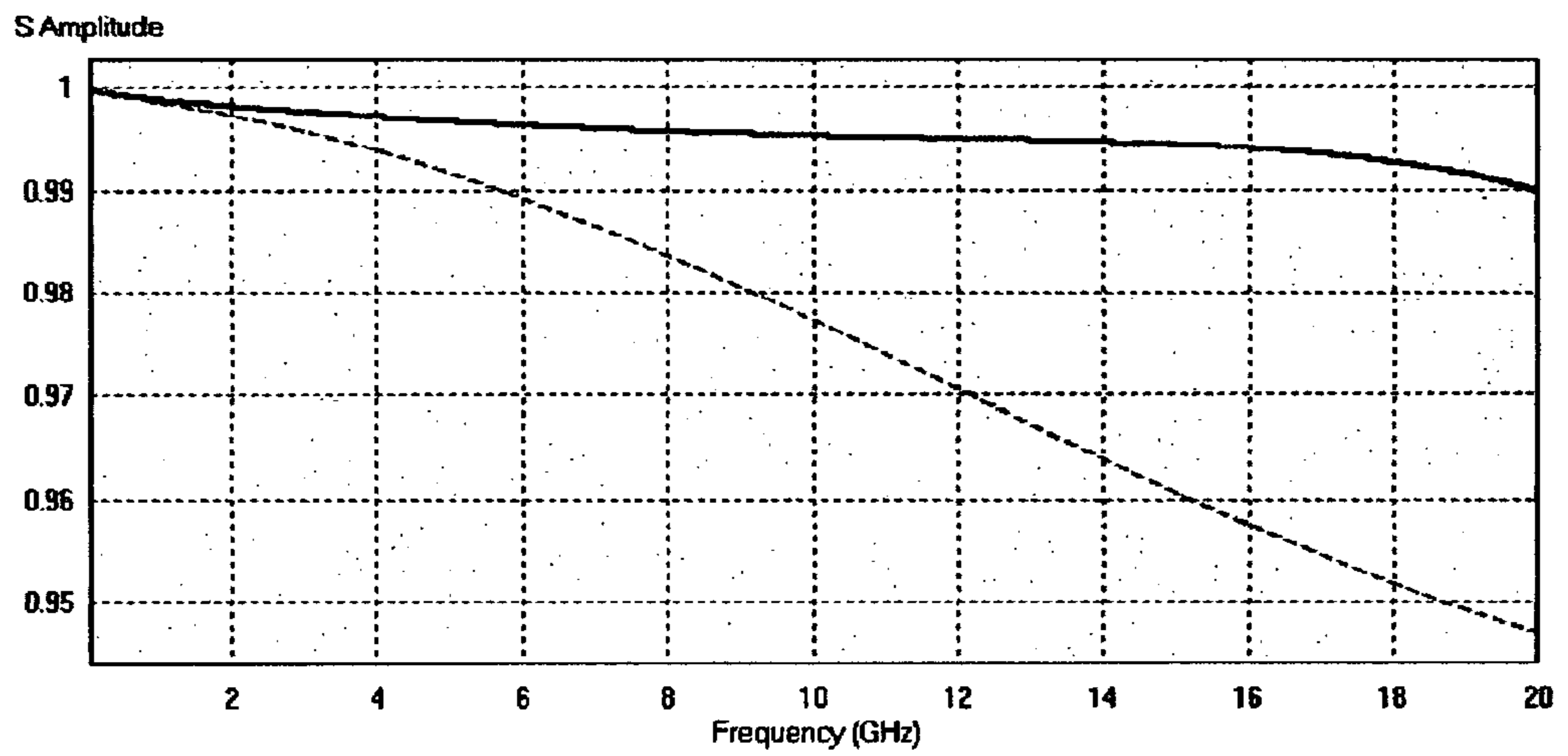


Figure 8

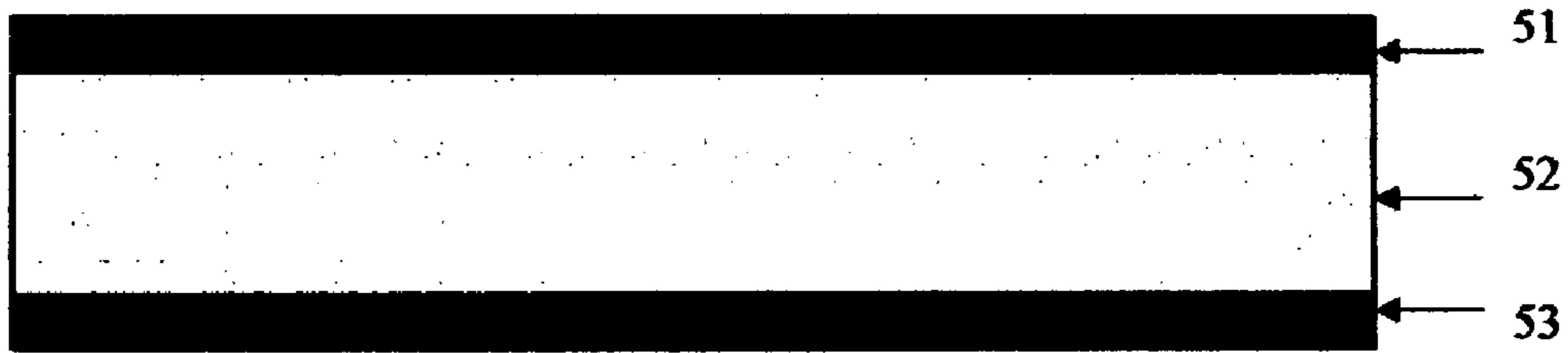


Figure 9a

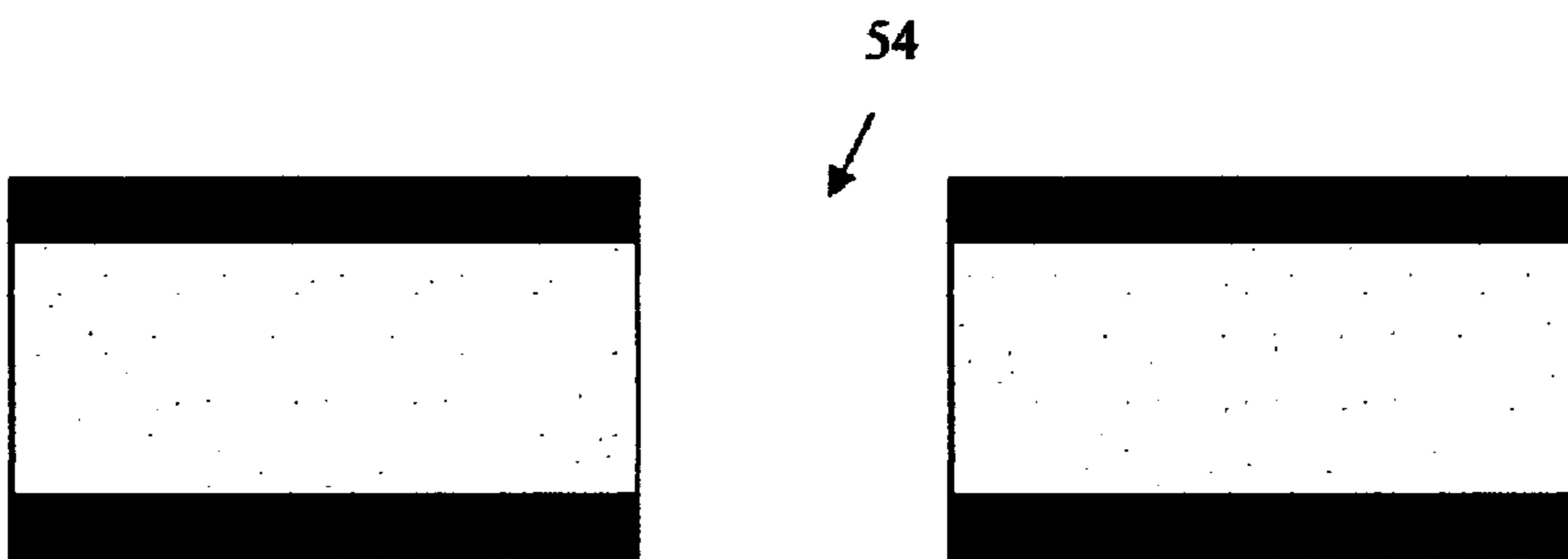


Figure 9b

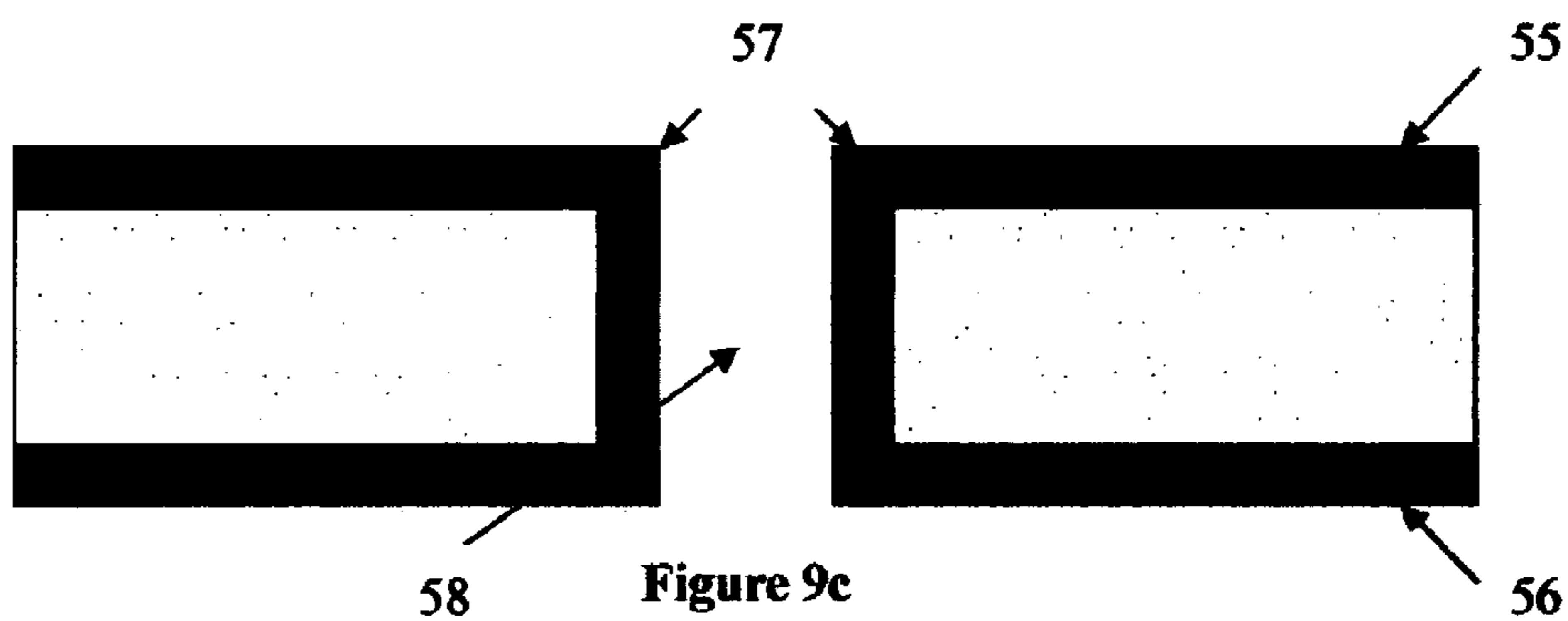


Figure 9c

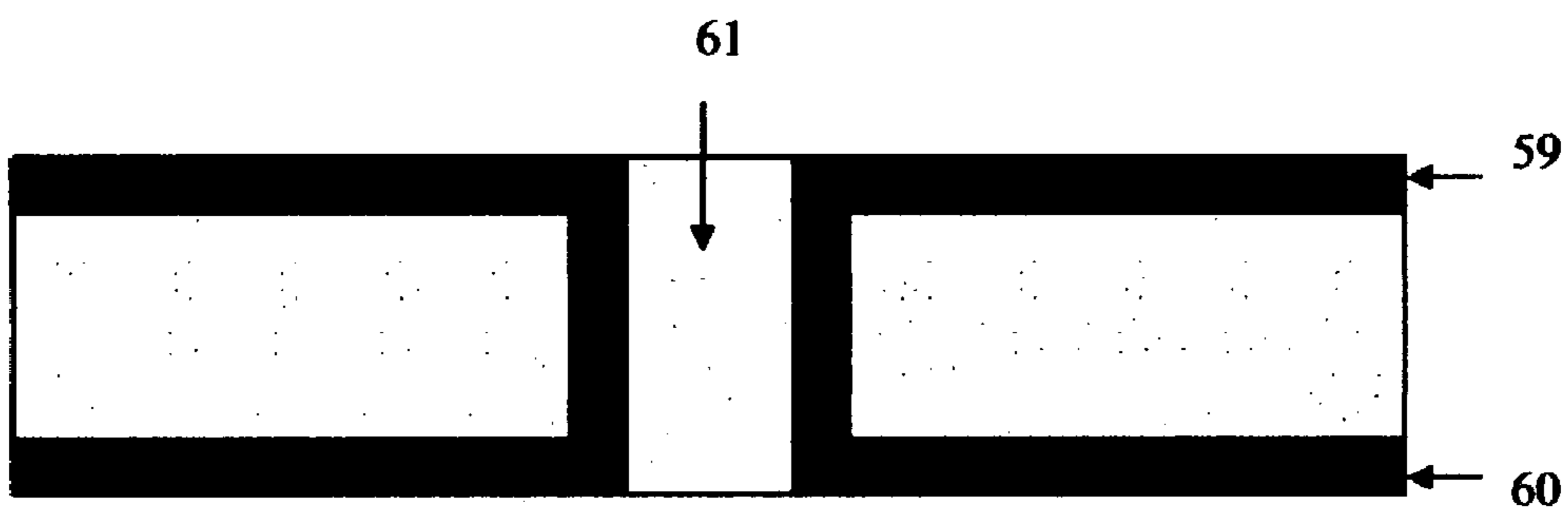


Figure 9d

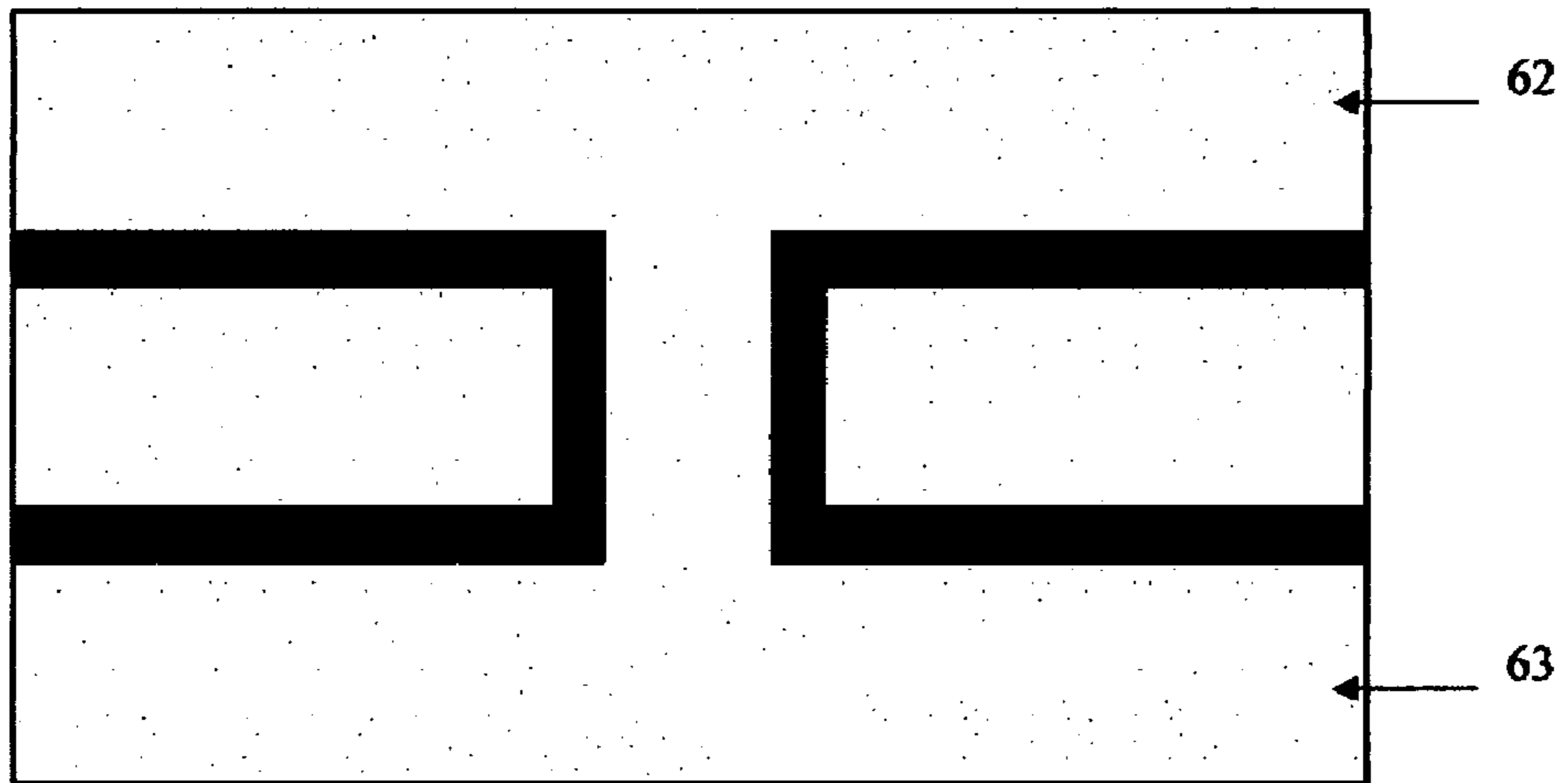


Figure 9e

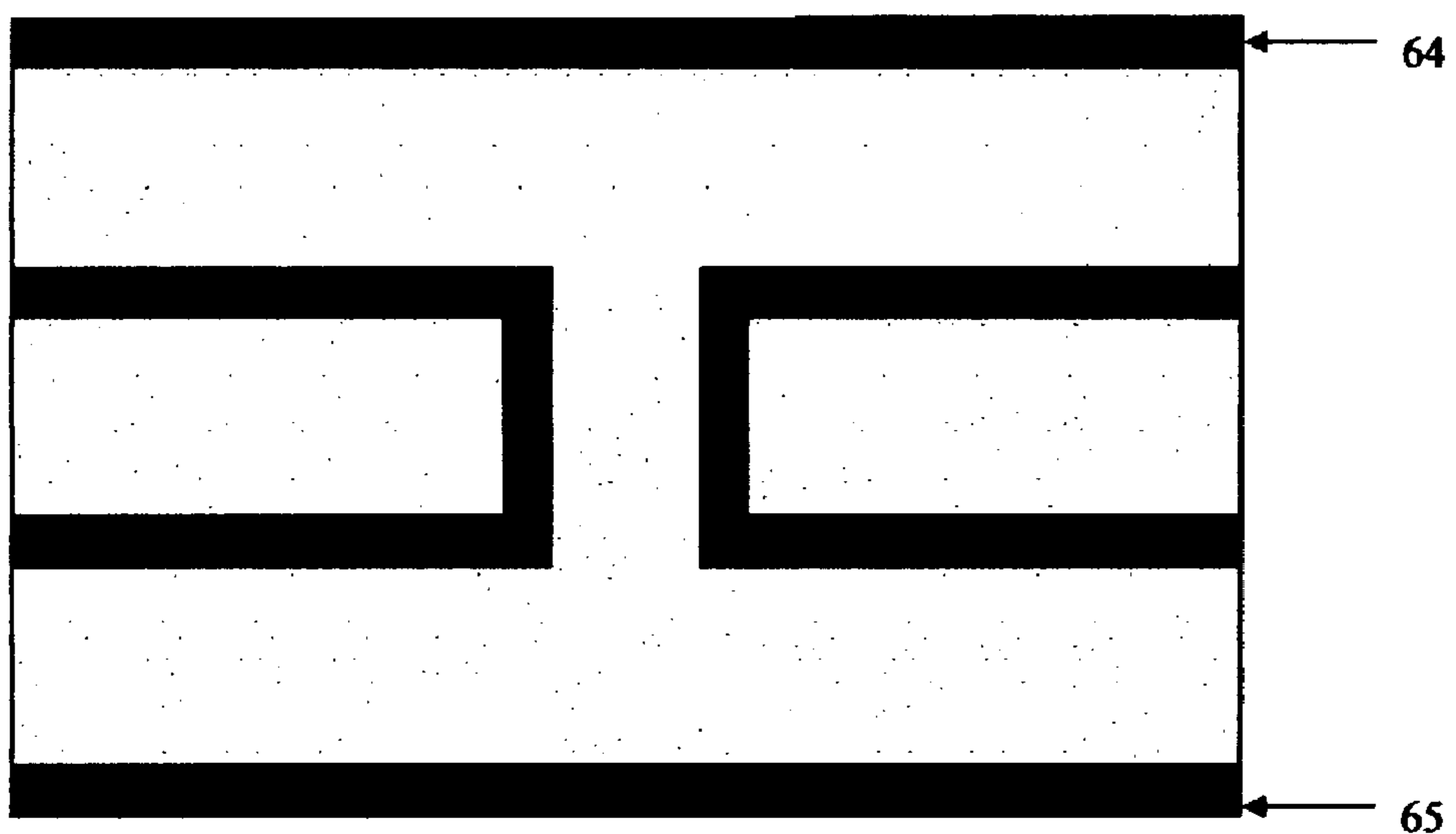


Figure 9f

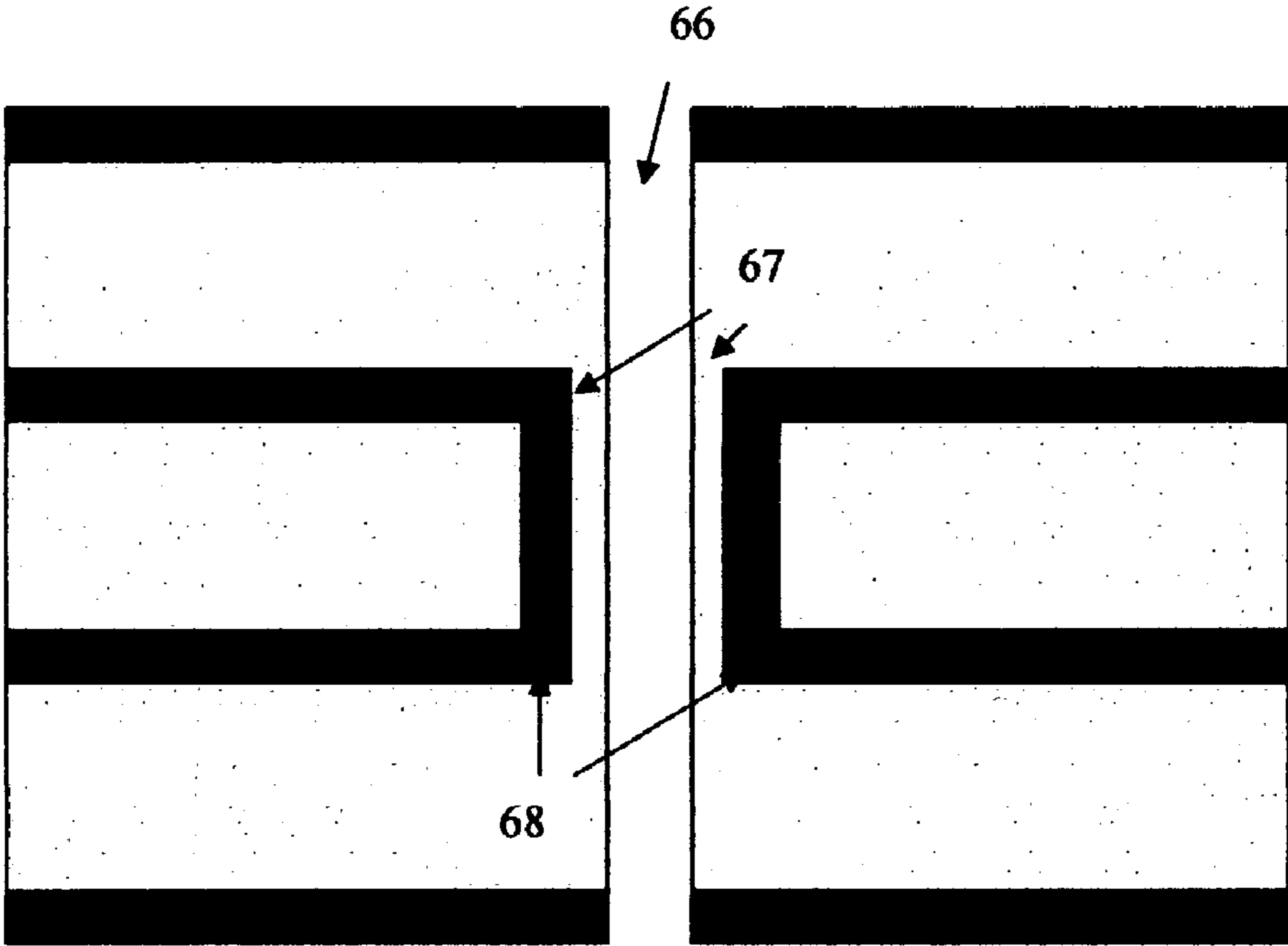


Figure 9g

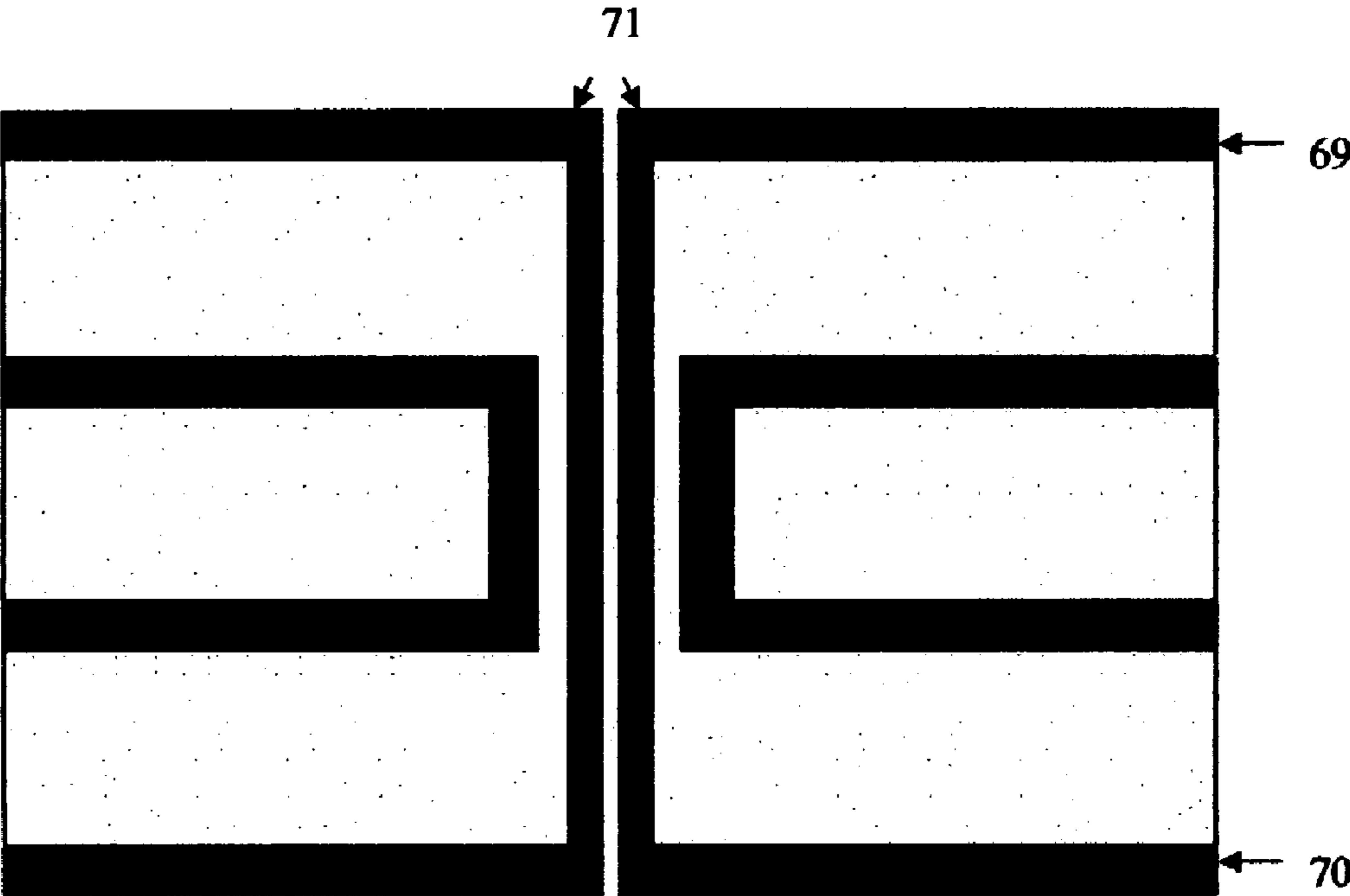


Figure 9h

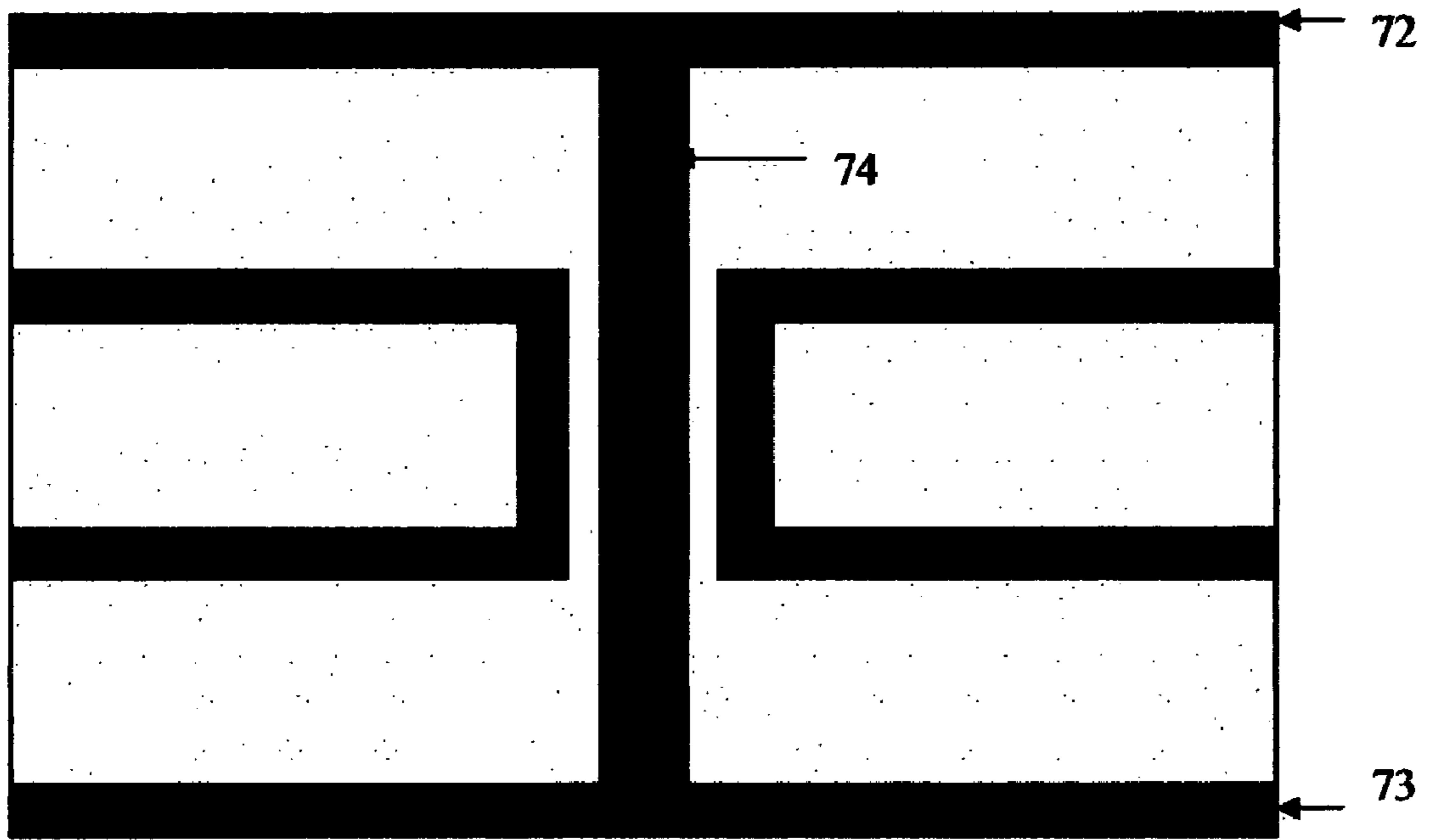


Figure 9i

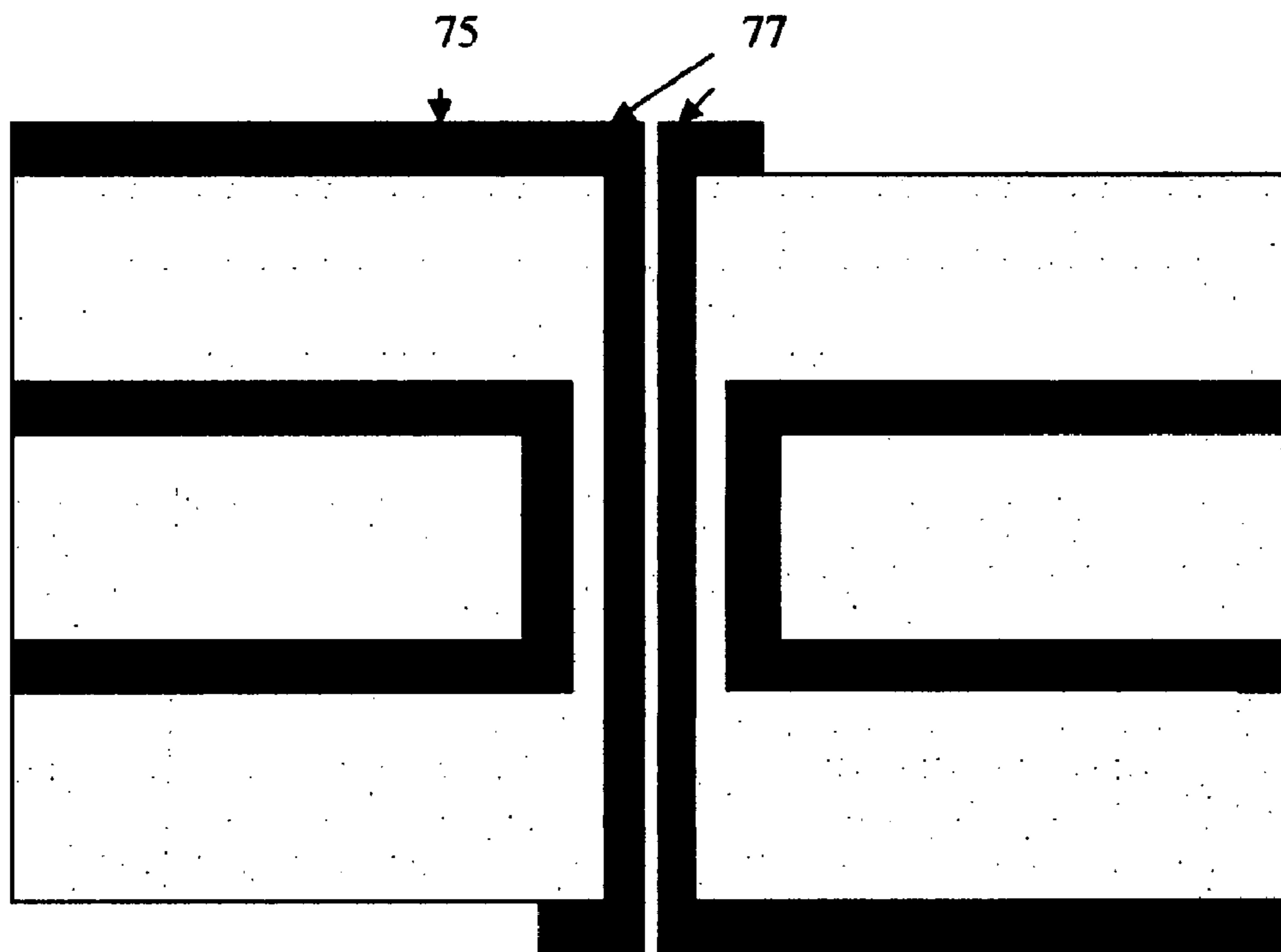


Figure 9j

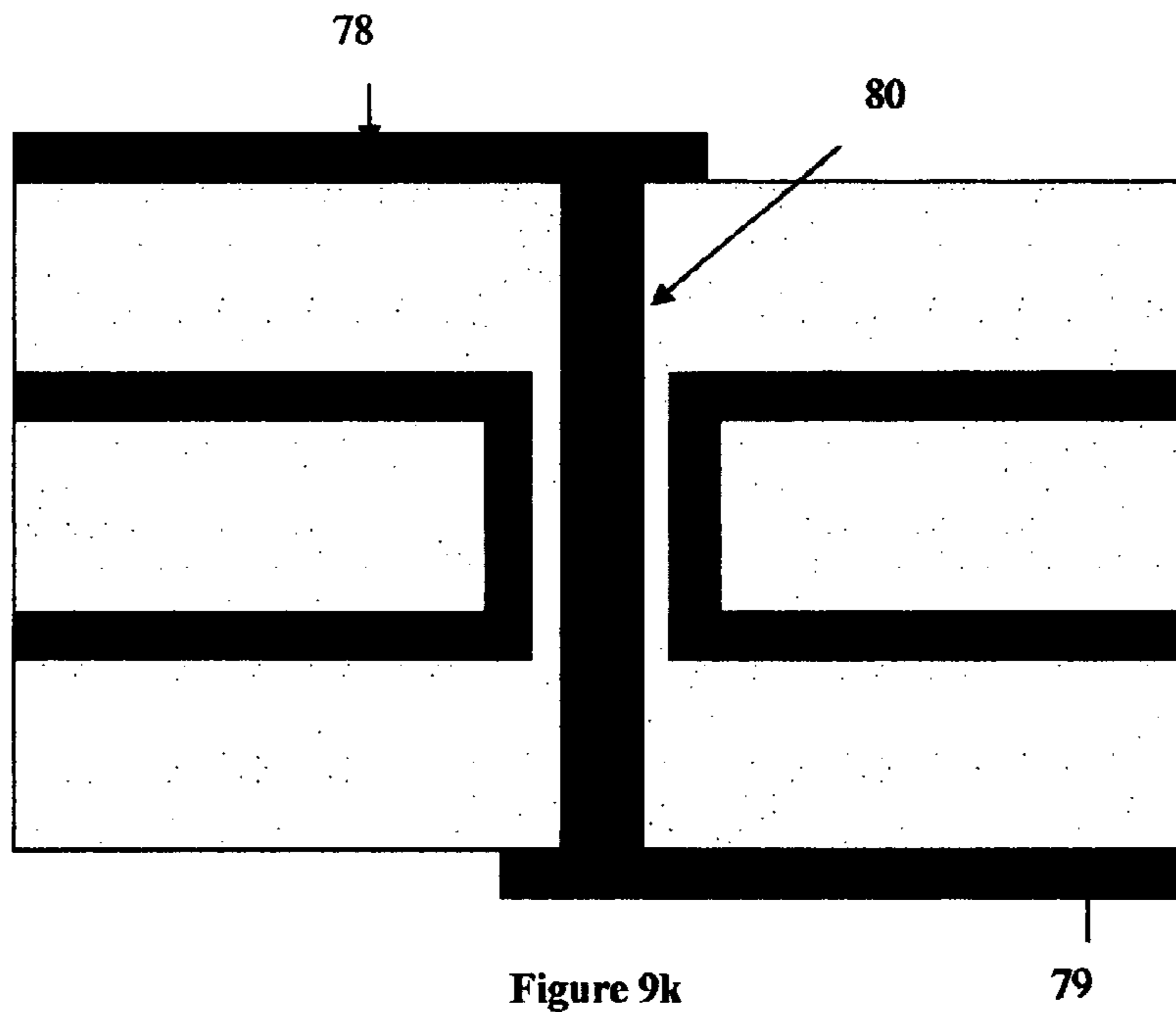


Figure 9k

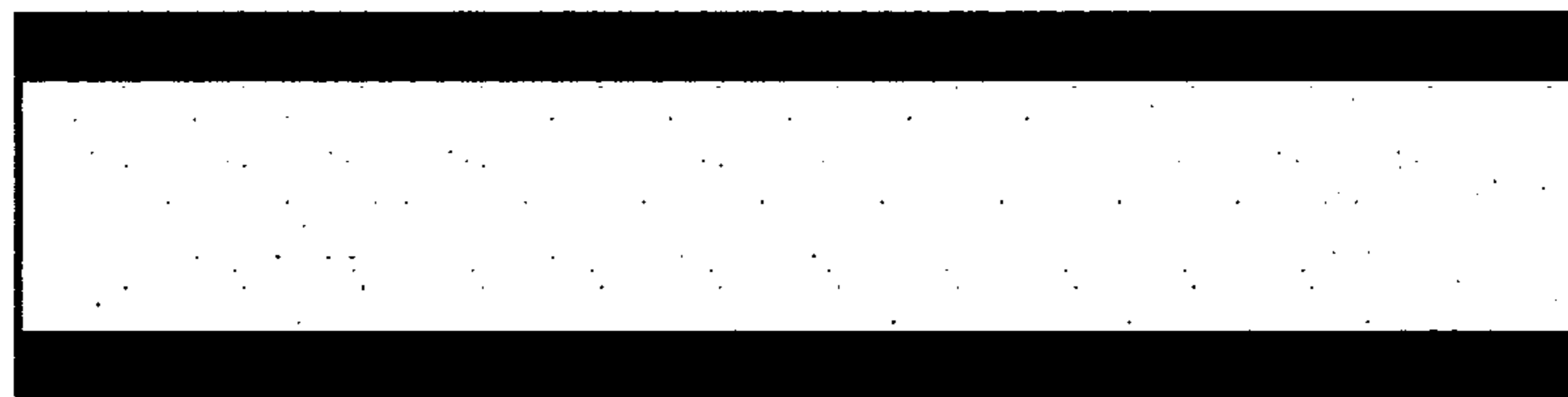


Figure 10a



Figure 10b



Figure 10c

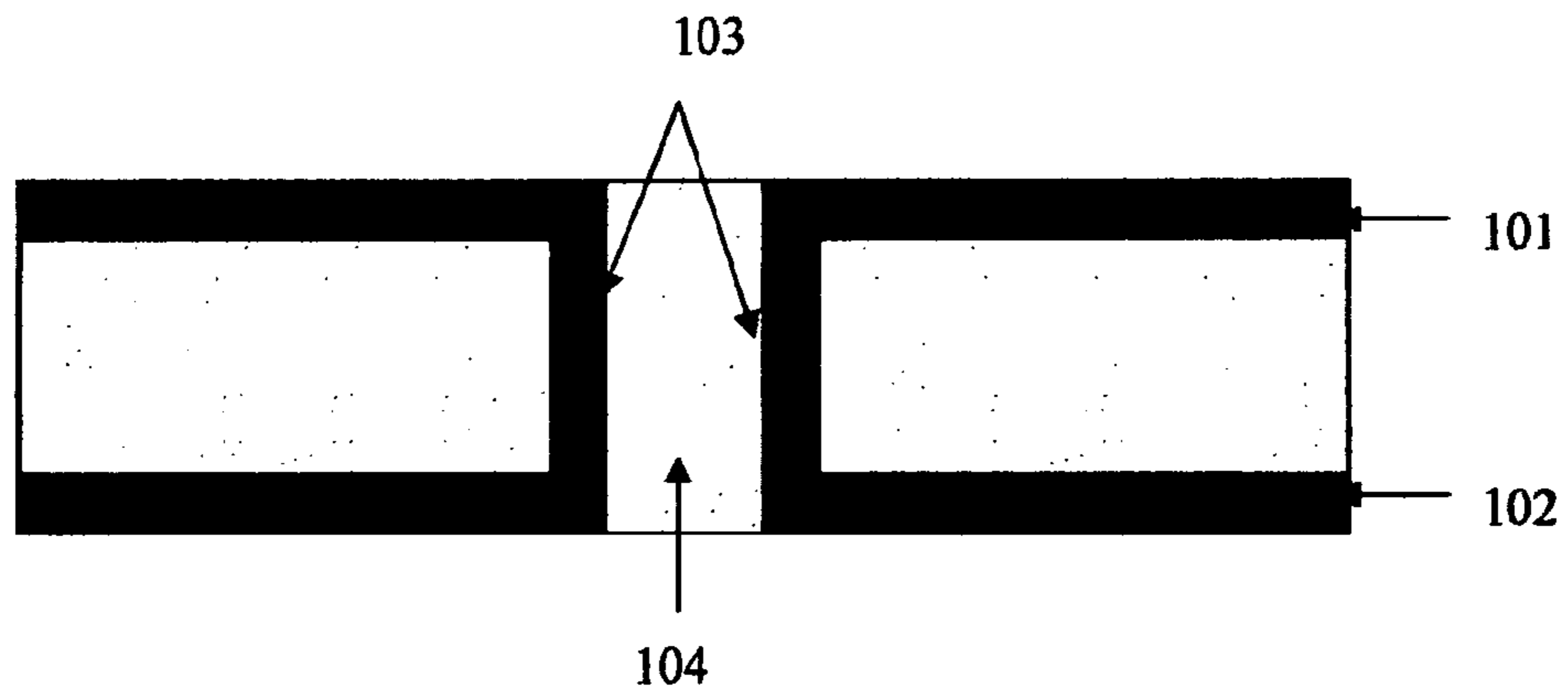


Figure 10d

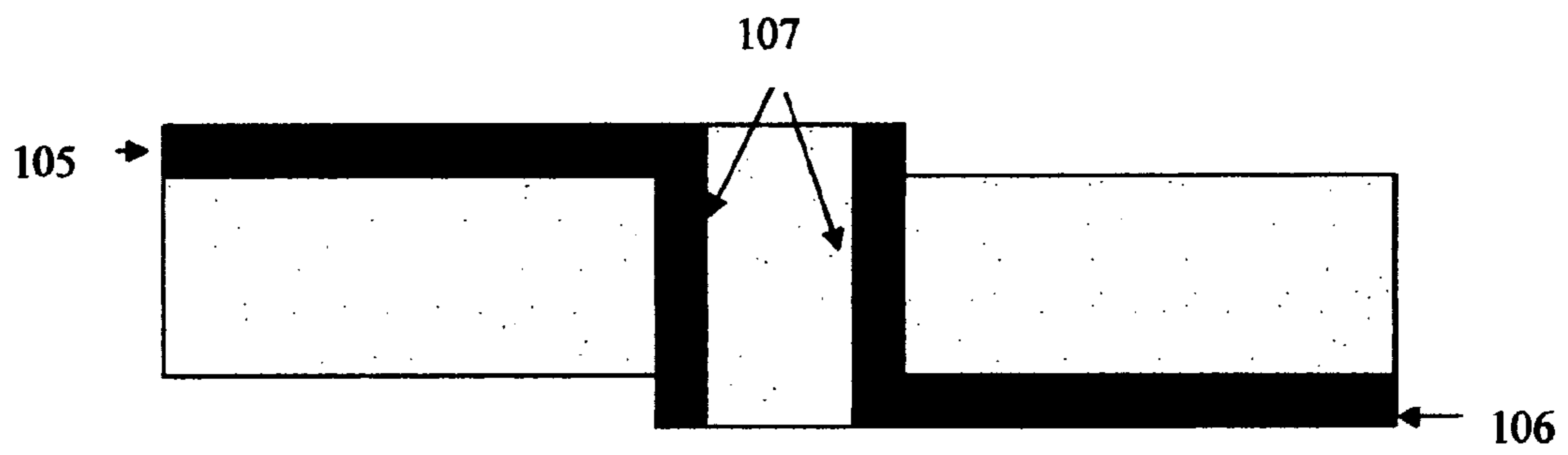


Figure 10e

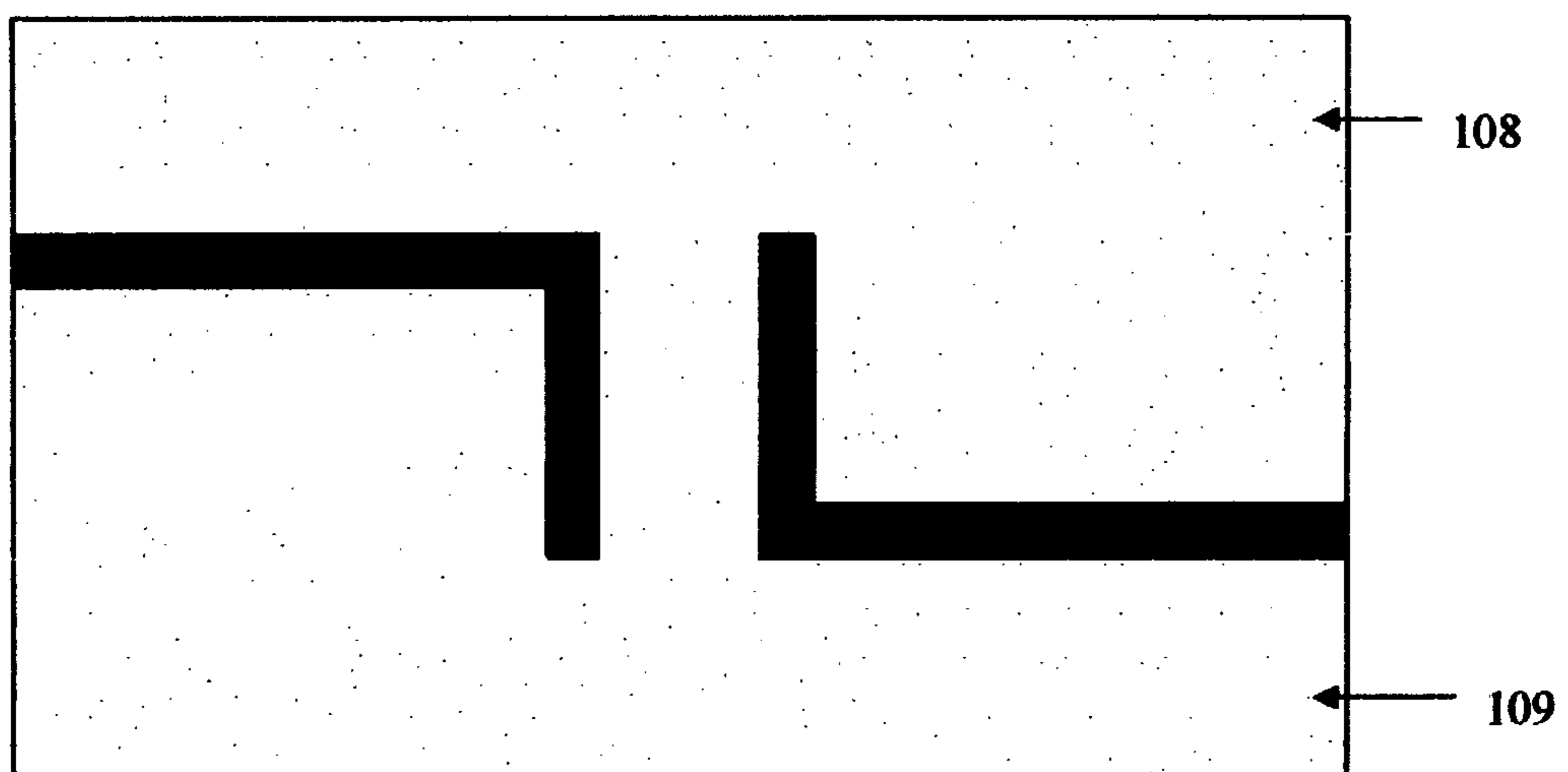


Figure 10f

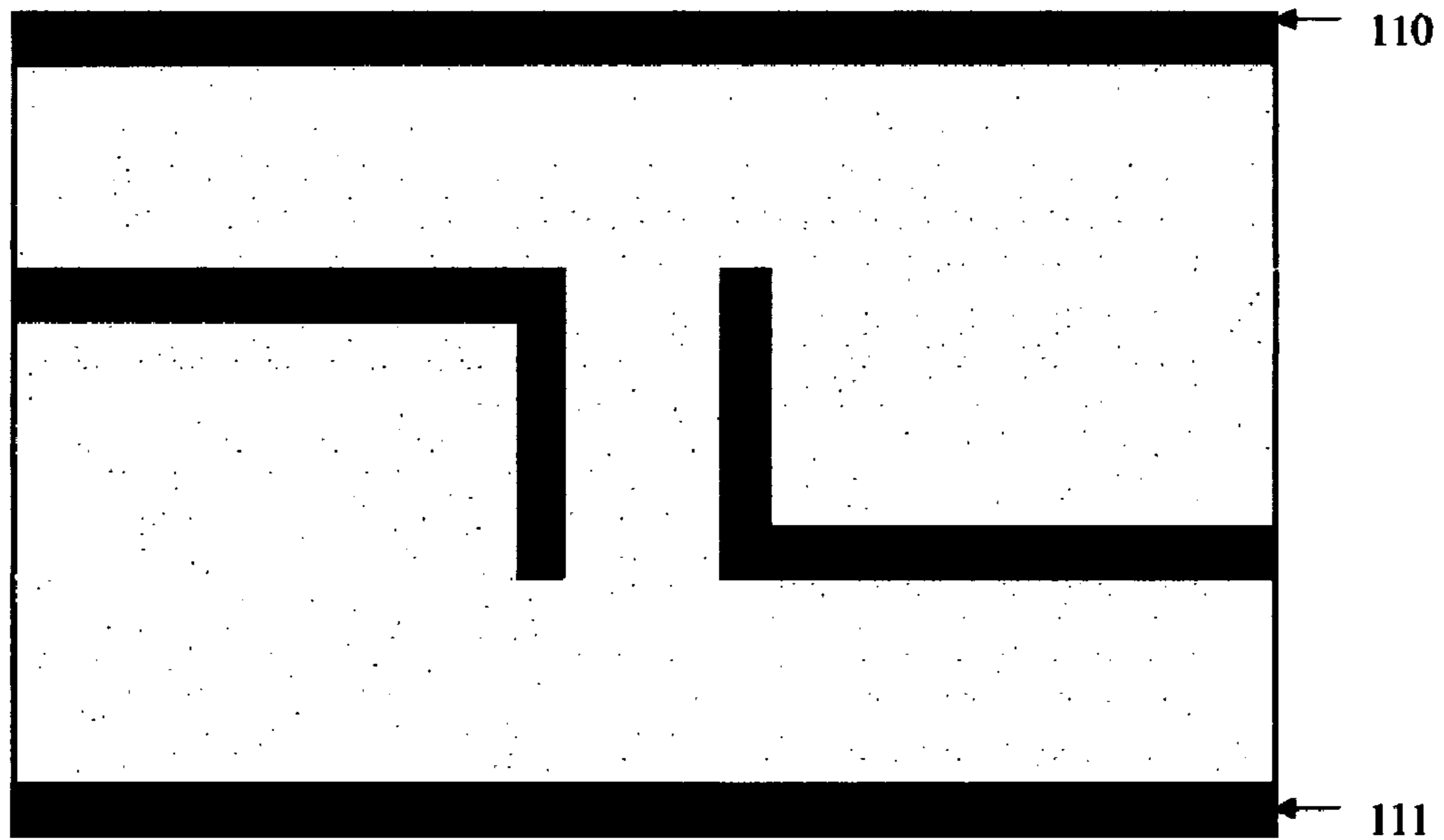


Figure 10g

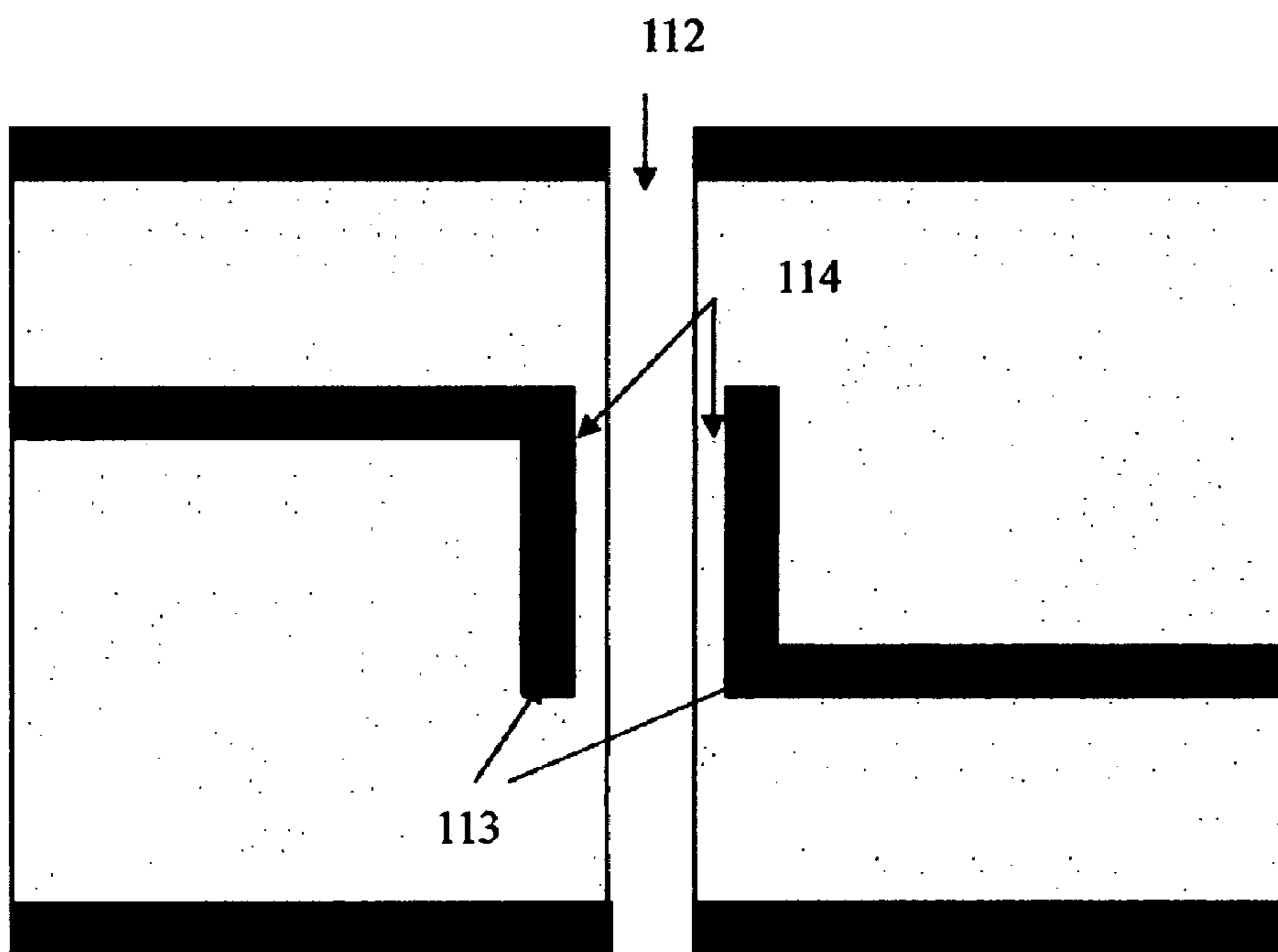


Figure 10h

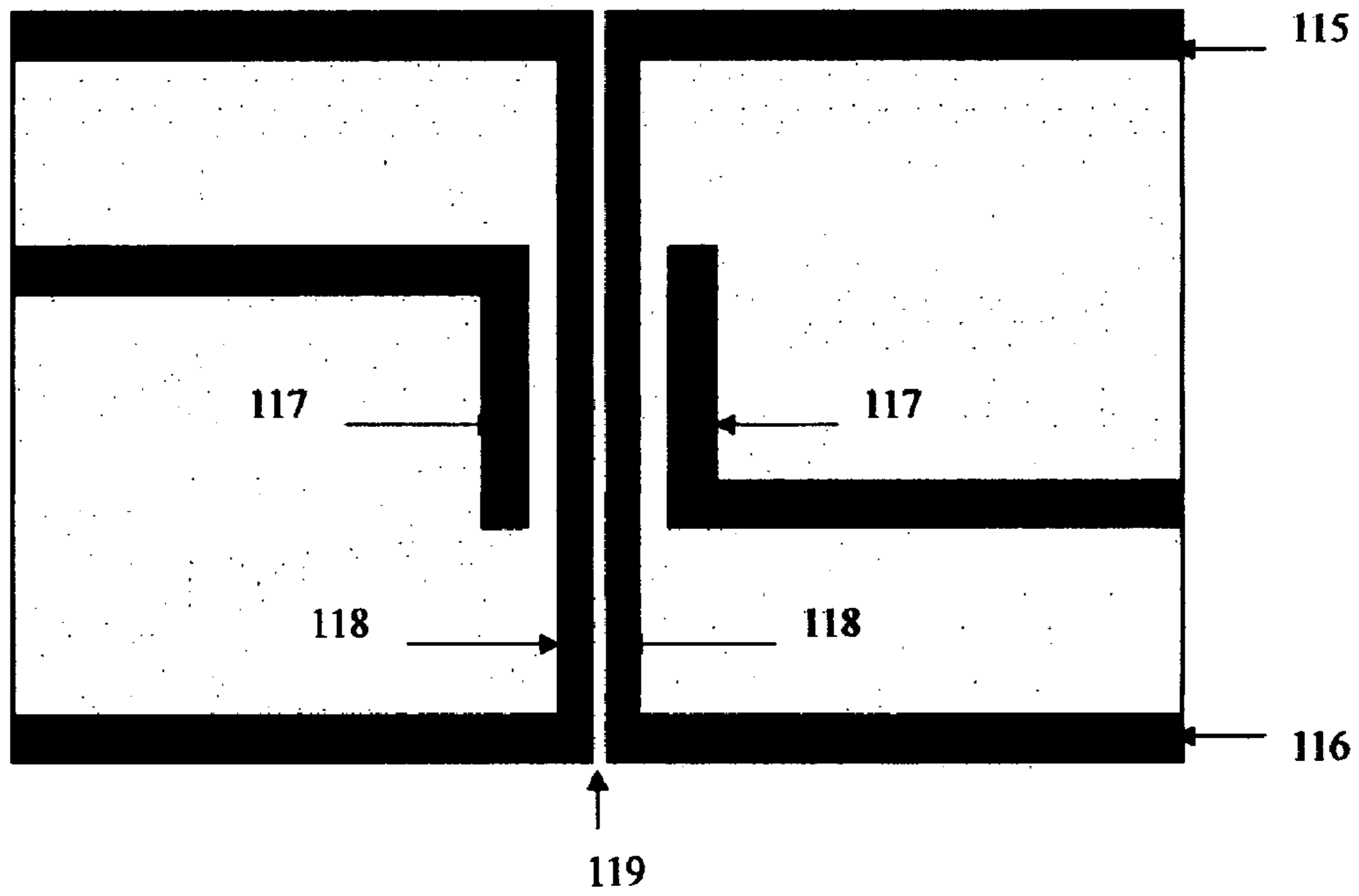


Figure 10i

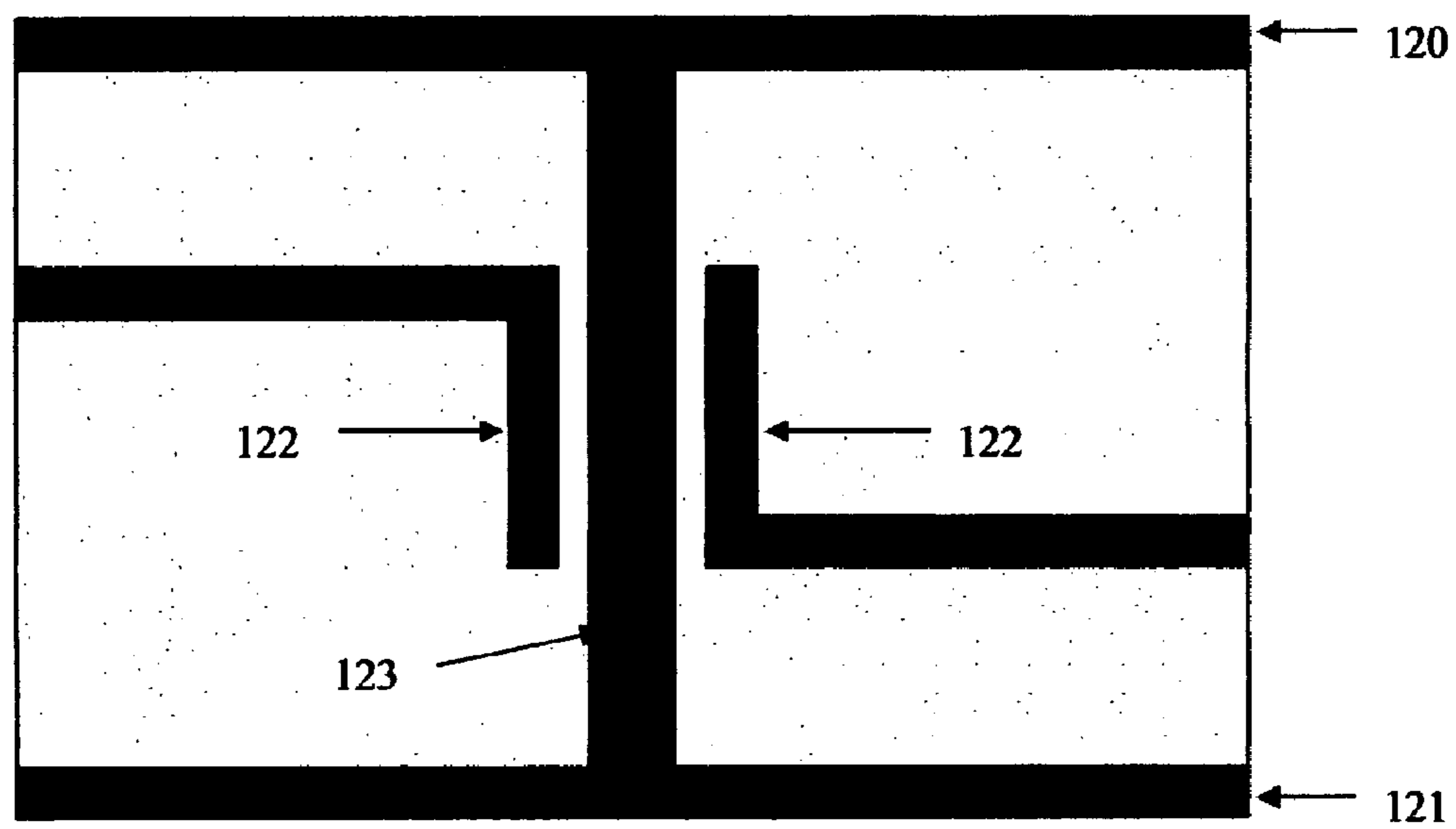


Figure 10j

VIA STRUCTURE FOR IMPROVING SIGNAL INTEGRITY

Matter enclosed in heavy brackets [] appears in the original patent but forms no part of this reissue specification; matter printed in italics indicates the additions made by reissue.

CROSS REFERENCE TO RELATED APPLICATIONS

The invention relates to an application that is entitled "Circuit Board Via Structure for High Speed Signaling," (Publication NO.: US 2006/0237227 A1), which is filed presently with the U.S. Patent & Trademark Office, and which is used herein for reference in its entirety.

FIELD OF THE INVENTION

The embodiments of this invention relate to printed circuit boards and integrated circuit packaging boards, and in particular to a novel via structure for improving the signal integrity and reducing the EMC/EMI problem. The novel via structure can be incorporated into a variety of integrated, as well as other types of circuits.

BACKGROUND

In multilayer printed circuit boards, integrated circuit packages, and integrated circuits on dies, there are many situations in which signals need to switch from layer to layer. FIG. 1 and FIG. 3 show two situations of signal switching. From FIG. 1, a signal trace 2 on the top layer switches to the signal trace 9 on the bottom layer through the signal via 3. The signal propagating along via 3 will pass through the two metal planes 7 and 8, which can be either a power plane or ground plane. When the signal propagates through via 3, it will generate an electromagnetic (EM) wave. The EM wave will propagate between the two metal planes 7 and 8, causing voltage fluctuations that will degrade the signal integrity. Also, the impedance of the signal propagating in a trace will be different from that of the signal propagating in a via. This is the so-called impedance discontinuity.

This impedance discontinuity will affect the signal quality as well. The dashed lines of FIGS. 5 and 7 show the simulated insertion loss (S_{21}) of the signal path indicated in FIG. 1 for the PCB board-level via and the IC packaging-level via, respectively. A smaller insertion loss value indicates that there is more energy dissipated along the signal path. These energy losses could be due to EM radiation, impedance discontinuity, dielectric loss, and so on.

From FIG. 3, a signal propagating along trace 23 on the third layer switches to a signal propagating along trace 30 on the fifth layer through the signal via 24. The signals on traces 23 and 30 will be referenced to the metal planes 28 and 29, respectively. The metal planes 28 and 29 can be power planes or ground planes. When the signal propagates along the signal via 24, it will also generate an EM wave. The EM wave will propagate between the two metal planes 28 and 29 and will cause voltage fluctuations that will degrade the signal integrity.

Again, the impedance along the signal trace will be different from that along the signal via. This impedance discontinuity will also affect the signal quality. The dashed lines of FIG. 6 and FIG. 8 show the simulated insertion loss (S_{21}) associated with the signal path indicated in FIG. 3 for the PCB

board-level via and the IC packaging-level via, respectively. Again, the smaller insertion loss values indicate that there is more energy dissipated along the signal path. These energy losses could be due to EM radiation, impedance discontinuity, dielectric loss, and so on.

SUMMARY

The embodiment of the invention is about a novel via structure which can be incorporated into printed circuit boards, integrated circuit packages, and integrated circuits in order to reduce crosstalk, to improve signal integrity and to achieve EM emission compliance. A 4-layer (2 signal layers and 2 power layers or 2 signal layers and 2 ground layers) circuit board assembly was used for demonstrating the effect of the novel via structure. The same concept can be applied to any multi-layer circuit board. Layers that have an electrical property can be added above, under, or within the basic 4-layer circuit board to achieve a multi-layer circuit board. For 2-layer and 3-layer circuit boards, a deformed version of the proposed via structure based upon the same concept will be needed for a coplanar waveguide configuration.

The 4-layer circuit board can be separated into 3 types of circuit structures.

Type A (see FIG. 2): one signal layer is on the top layer; 2 ground layers or 2 power layers represent the two middle layer; and the other signal layer is on the bottom layer. The signal via connects the transmission lines that lie on the top and bottom layers. The ground via or power via is plated surrounding the signal via, i.e. the ground via or power via is concentric to the signal via. The plated ground via or the power via needs to be kept a non-trivial distance (such as the diameter of the anti-pad) from the signal via to avoid shorting. The impedance of the signal via can be properly designed by adjusting this distance between the signal via and the ground via (or power via). The thickness of the plated ground via (or power via) must be larger than the skin depth at the frequency of the signal. It is assumed that copper will be used for the ground via or power via in this implementation. The plated via connects the third layer of the ground plane (or power plane) to the fifth layer of the ground plane (or power plane) to be ground via (or power via). Note that it is necessary that the third layer and the fifth layer have the same electrical properties. With this configuration the signal via can be isolated to prevent electromagnetic coupling with other vias (not shown in the figure) on the circuit board. Also, the plated ground via (or power via) provides a voltage reference or a complete current return path for the signal propagating through the signal via. Eventually, the impedance discontinuity between the transmission line and the signal via will be reduced so that the subsequent reflection effect and, hence, the return loss will be minimized. Due to the separation of the signal vias by using the plated ground via (or power via), the crosstalk between signal vias will also be reduced to improve the quality of the signals. Consequently, the radiation on the edge of the circuit board (EMC/EMI problem) will be minimized as well.

Type B (see FIG. 4): the top layer and bottom layer are ground layers or power layers. The third layer and the fifth layer are signal layers. The signal via connects the transmission line on the third layer with the transmission line on the fifth layer. A ground via or power via is created to connect the top layer with the bottom layer. The diameter of the ground via or power via is smaller than that of the signal via to avoid shorting. The impedance of the signal via can be properly designed by controlling the distance between signal via and the ground via (or power via) as well. The ground via or power

3

via is concentric to the signal via. Note that it is necessary that the top layer and the bottom layer have the same electrical properties. The thickness of the plated ground via or power via is larger than the skin depth associated with the frequency of the signal. The ground via or power via is assumed to be a highly conductive metal, such as copper, in this implementation. Also, the ground via (or power via) provides a voltage reference plane or a complete current return path for the signal propagating through the signal via. Finally, the impedance discontinuity between the transmission line and the signal via will be reduced so that the subsequent reflection effect and, hence, the return loss will be minimized. Consequently, the radiation on the edge of the circuit board (EMC/EMI problem) will be minimized as well.

Type C (see FIG. 2 or FIG. 4): Refer to Type A or Type B. The signal layers are replaced with power layers or ground layers. The resulting configurations, i.e., the stack from the top to the bottom layers, turn out to have the generic structure: power-ground-ground-power or ground-power-power-ground. When the signal via is replaced with a power via or a ground via, the via structure will become a "capacitor", an element that can store electrical energy. The value of the capacitor can be properly designed by filling the space between the power via and ground via with a material having a selected permittivity value, or by adjusting the distance between the power via and ground via. The addition of a capacitor will benefit the power integrity because the voltage fluctuations between the power plane and the ground plane can be minimized. Finally, the overall signal integrity of the system will be improved due to the reduction of the voltage fluctuations in these layers.

BRIEF DESCRIPTION OF THE DRAWING

FIG. 1 shows a prior microstrip-like signal via structure without a power via or a ground via

FIG. 2 shows a novel microstrip-like signal via surrounded by a power via and a ground via

FIG. 3 shows prior stripline-like signal via without a power via or a ground via

FIG. 4 shows a novel stripline-like signal via surrounding the power via and the ground via

FIG. 5 shows the simulated insertion loss (S_{21}) of the PCB board-level size of the microstrip-like via. The dashed line represents the magnitudes of the simulated S_{21} values predicted for the structure shown in FIG. 1: The solid line represents the magnitudes of the simulated S_{21} values predicted for the structure shown in FIG. 2.

FIG. 6 shows the simulated insertion loss (S_{21}) of the PCB board-level size of the stripline-like via. The dashed line represents the magnitudes of the simulated S_{21} values predicted for the structure shown in FIG. 3. The solid line represents the magnitudes of the simulated S_{21} values predicted for the structure shown in FIG. 4.

FIG. 7 shows the simulated insertion loss (S_{21}) of the IC packaging-level size of the microstrip-like via. The dashed line represents the magnitudes of the simulated S_{21} values predicted for the structure shown in FIG. 1. The solid line represents the magnitudes of the simulated S_{21} values predicted for the structure shown in FIG. 2.

FIG. 8 shows the simulated insertion loss (S_{21}) of the IC packaging-level size of the stripline-like via. The dashed line represents the magnitudes of the simulated S_{21} values predicted for the structure shown in FIG. 3. The solid line represents the magnitudes of the simulated S_{21} values predicted for the structure shown in FIG. 4.

4

FIGS. 9a-9k show the sequential fabrication steps required for the formation of the novel via structure shown in FIG. 2.

FIGS. 10a-10j show the sequential fabrication steps required for the formation of the novel via structure shown in FIG. 4.

TABLE 1

The elements of FIG. 1	
Printed circuit board (PCB) or substrate or die	1
The first signal trace in first layer	2
The signal via	3
The first medium in the second layer	4
The second medium in the fourth layer	5
The third medium in the sixth layer	6
The first power plane or ground plane in the third layer	7
The second power plane or ground plane in the fifth layer	8
The second signal trace in the seventh layer	9
Anti-pad	10

TABLE 2

The elements of FIG. 2	
Printed circuit board (PCB) or substrate or die	11
The first signal trace in the first layer	12
The signal via	13
The power via (or ground via)	14
The first medium in the second layer	15
The second medium in the fourth layer	16
The third medium in the sixth layer	17
The first power plane (or ground plane) in the third layer	18
The second power plane (or ground plane) in the fifth layer	19
The second signal trace in the seventh layer	20
Anti-pad	21

TABLE 3

The elements of FIG. 3	
Printed circuit board (PCB) or substrate or die	22
The first signal trace in the third layer	23
The signal via	24
The first medium in the second layer	25
The second medium in the fourth layer	26
The third medium in the sixth layer	27
The first power plane or ground plane in the first layer	28
The second power plane or ground plane in the seventh layer	29
The second signal trace in the fifth layer	30

TABLE 4

The elements of FIG. 4	
Printed circuit board (PCB) or substrate or die	31
The first power plane (or ground plane) in the first layer	32
The signal via	33
The power via (or ground via)	34
The first medium in the second layer	35
The second medium in the fourth layer	36
The third medium in the sixth layer	37
The first signal trace in the third layer	38
The second signal trace in the fifth layer	39
The second power plane (or ground plane) in the seventh layer	40
Anti-pad	41

DETAILED DESCRIPTION

The purpose of the invention is to provide a novel via structure that can reduce via-to-via crosstalk, the impedance discontinuity, and EMC radiation. In particular, the signal

5

integrity will be improved and the edge radiation of the PCB board or IC packaging will be minimized.

A 4-layer (2 signal layers and 2 power layers or 2 ground layers) circuit board with a single via structure is used for demonstrating the interest of the invention. The 4-layer circuit board is comprised of 2 signal layers, 3 medium layers, 2 power layers (or 2 ground layers), and a via structure. The signal via electrically connects the signal traces on the different layers. The power via or ground via could be built outside the signal via (see FIG. 2) or inside the signal via (see FIG. 4). For the power via (or ground via) built outside the signal via, the thickness of the power via (or ground via) must be larger than the skin depth associated with the frequency of the signal on the signal via. Consequently, the EM wave generated by the signal via will not penetrate through the power via (or ground via) and, as a result, the via-to-via crosstalk will be mitigated. Also, the power via (or ground via) will provide the signal a shortest current return path and will reduce the impedance discontinuity. This in turn will minimize the return loss and, hence, will improve the signal integrity. For the power via (or ground via) built inside the signal via, the thickness of the power via (or ground via) must also be larger than the skin depth associated with the frequency of the signal on the signal via to reduce the losses in the metal. The power via (or ground via) will provide the signal a shortest current return path and will reduce the impedance discontinuity to minimize the return loss and improve the signal integrity.

Four circuit structures (FIG. 1-FIG. 4) are used to explain the concept of the invention. Each circuit structure comprises 2 signal layers, 3 medium layers, 2 power layers (or 2 ground layers), and a via structure. The concept of the invention can be applied to integrated circuit packaging, integrated circuits on a die, or many other types of circuits.

FIG. 1 is a prior via structure implemented on the circuit board 1. There is no power via or ground via between the power planes (or ground planes). Therefore, when the signal via 3 passes through the power planes (or ground planes), the signal will generate EM waves between the planes and these EM waves will fluctuate the voltages on the various planes. Moreover, there is a discontinuity between the impedances associated with the signal traces and the signal via. The subsequent reflected signals will also affect the signal quality.

FIG. 2 is used for demonstrating the implementation of the Type A version of the invention. The novel via structure alleviates the factors which adversely affect the signal integrity such as via-to-via coupling, impedance discontinuity, EM radiation, and so on. The circuit board is comprised of a substrate 11; two signal layers 12 and 20; two power layers (or two ground layer) 18 and 19; a signal via 13; and a power via (or a ground via) 14. The signal layer 12 is electrically connected to the signal layer 20 by the signal via 13. The power via (or ground via) 14 electrically connects the two power layers (or the two ground layer) 18 and 19; it is plated and surrounds the signal via 13. The power via (or ground via) 14 will provide a complete voltage reference surface and the shortest current return path for the signal that propagates along the signal via 13. Also, the power via (or ground via) 14 will isolate any electromagnetic (EM) wave generated by the signal via 13 and, hence, it will prevent any EM couplings with other vias (not shown).

The via structures that passes through the substrate 11 can be formed by mechanical Or laser drilling. The via hole can be a "blind hole" or "through hole". First, the power blind via (or ground blind via) 14 is formed in order to connect the two power layers (or the two ground layers). The material of the power via (or ground via) can be any highly conductive alloy or metal such as Tin (Sn), silver (Ag), copper (Cu), gold (Au).

6

The signal layer 12 is then electrically connected with the signal layer 20 through the signal via 13. Any techniques such as casting, plating, or non-plating can be adapted for the formation of any via in the embodiments of the invention.

The power via (or ground via) 14 is plated with copper in this example of the invention. The signal via 13 and the power via (or ground via) 14 are circular and concentric. However, they do not necessarily have to be circular or concentric. For example, the signal via 13 can be a solid cylinder or a hollow cylinder non-centric with the power via (ground via) 14. Also, the signal via 13 must be kept at least a distance equal to the size of the anti-pad from the power via (or ground via) 14. The thickness of the power via (or ground via) 14 must be larger than the "skin depth" associated with the frequency of the signal propagating along the signal via 13. The skin depth describes the fact that any current flowing in a conductor will migrate toward the surface or "skin" of the conductor when the signal frequency gets higher. As a result, most of the current will be concentrated within the skin depth of the conductor. The skin depth $\delta_s = (2/\omega\mu\sigma)^{1/2}$, where ω is the frequency of the signal, μ is the permeability of the conductor, and σ is the conductivity of the conductor. Therefore, if the conductor is thick enough, the EM wave generated by the signal via will not penetrate through the power via (or ground via). As a result, the insertion loss (S_{21}) due to radiation emitted by the signal via will be minimized and, consequently, the via-to-via coupling will be reduced. Therefore, the signal integrity will be improved.

FIG. 3 is the other prior via structure. It is shown on the circuit board 22. There is no power via (or ground via) between the power planes (or ground planes) 28 and 29. Therefore, when the signal via 24 passes through the power planes (or ground planes) 28 and 29, the signal propagating along it will generate EM waves between these planes and these EM waves will cause fluctuations in the voltage on those planes. Also, the impedance between the signal traces and the signal via is discontinuous so there will be reflected signals and they will affect the signal quality.

FIG. 4 is used for the demonstration of the implementation of the Type B version of the invention. This novel via structure also alleviates the factors of affecting signal integrity such as via-to-via coupling, impedance discontinuity, EM radiation, and so on. The circuit board comprises a substrate 31; two signal layers 38 and 39; two power layers (or two ground layers) 32 and 40; a signal via 33; and a power via (or a ground via) 34. The signal layer 38 is electrically connected to the signal layer 39 with the signal via 33. The signal via 33 that electrically connects the two signal layers 38 and 39 is plated and surrounds the power via (or ground via) 34. The power via (or ground via) 34 will provide a complete voltage reference surface and the shortest current return path for the signal propagating along the signal via 33. Also, any EM waves generated by the signal via 33 will be concentrated between the power via (or ground via) 34 and the signal via 33 so that the signal via 33 will couple less energy with other vias (not shown).

Again, the via structures passing through the substrate 31 can be formed by mechanical or laser drilling. The via hole can be a "blind hole" or a "through hole". First, the signal blind via 33 is formed to connect electrically the signal layer 38 and the signal layer 39. The power through via (or ground through via) 34 then is formed to connect the two power layers (or the two ground layers) 32 and 40. The power via (or ground via) 34 can be plated to form a circular annular tube or filled with copper to form a solid cylinder. The material of the power via (or ground via) can be any conductive alloy or metal such as Tin (Sn), silver (Ag), copper (Cu), gold (Au).

Any standard technique such as casting, plating, or non-plating can be adapted for the formation of any via in the embodiments of the invention.

The signal via **33** is plated with copper in this example. The signal via **33** and the power via (or ground via) **34** are circular and concentric. However, it is not necessary that they be a circular annular tube or that they be concentric. For example, the power via (ground via) **34** can be a cylindrical ring, solid cylinder, or polygon ring which can be concentric or not with the signal via **33**. Also, the signal via **33** must be kept at least a distance equal to the size of the anti-pad from the power via (or ground via) **34**. The thickness of the power via (or ground via) **34** must be larger than the "skin depth" associated with the frequency of the signal propagating along the signal via **33**. Therefore, most of the return current induced by the signal via will propagate along the power via (or ground via) **34** instead of propagating partially along the power via (or ground via) **34** and partially through the medium (not shown). The insertion loss (S_{21}) due to the metal losses in the power via (or ground via) will be minimized. The via-to-via EM coupling will be reduced as well. Finally, the signal integrity will be improved.

The construction of the novel via structure shown in FIGS. **2** and **4** can be accomplished using typical PCB fabrication techniques. FIGS. **9a-9k** show the sequential steps required for the fabrication of the novel via structure shown in FIG. **2**. FIGS. **10a-10j** show the required sequential steps for the fabrication of the novel via structure shown in FIG. **4**.

FIG. **9a** shows the dielectric substrate **52** plated with conductive metal planes on both sides, **51** and **53**. The conductive metal planes will be used as power planes or ground planes. The material, FR4 is commonly used as the dielectric material for PCB boards. Any proper dielectric material can be used for these printed circuit boards, including Low Temperature Co-fired Ceramics (LTCC) or Rogers Duroid™.

FIG. **9b** shows that a through hole **54** is formed. Mechanical or laser drilling can be used for the construction of the through hole.

FIG. **9c** shows that the through hole **58** is plated with conductive metal **57** to connect electrically with the planes **55** and **56**.

FIG. **9d** shows that the plated through hole **58** in FIG. **9c** is filled with dielectric material **61**. The dielectric material **61** can be properly optimally selected to control the impedance of the signal via. Chemical vapor deposition can be applied to fill the hole. Many other possible techniques can be applied as well. The details of those well-known techniques are not discussed here.

FIG. **9e** shows that two sheets of dielectric material **62** and **63** are stuck to both sides of the substrate. To adhere the dielectric material with the substrate, the processes of heating and hardening will be needed.

FIG. **9f** shows that the top and bottom sides of the substrate are constructed with conductive metal **64** and **65**. Chemical vapor deposition or other known techniques can be used for depositing the conductive metal.

FIG. **9g** illustrates how the through hole **66** is constructed. Mechanical or laser drilling can be used for the construction of this through hole. The gap (anti-pad) **67** between the through hole **66** and the plated via **68** can be properly designed to control the impedance of the signal via.

FIG. **9h** shows that the through hole **66** in FIG. **9g** is plated with conductive metal **71** to connect the conductive metal **69** and **70** on the top and the bottom sides. The through hole **66** in FIG. **9g** can be also filled with conductive metal **74** as shown in FIG. **9i** to connect the conductive metal **72** and **73** on the top and the bottom sides, respectively.

FIG. **9j** shows that the conductive metal **69** and **70** in the FIG. **9h** is formed into signal traces **75** and **76**. The formation of the signal traces **75** and **76** can be achieved by using typical PCB fabrication techniques. Finally, the signal traces **75** and **76** are connected electrically with the conductive metal **77**. Again, FIG. **9k** shows that the conductive metal **72** and **73** in the FIG. **9i** is formed into signal traces **78** and **79**. The formation of the signal traces **78** and **79** can be achieved by using typical PCB fabrication techniques as well. Eventually, the signal traces **78** and **79** are connected electrically with the conductive metal **80**.

FIGS. **10a-10d** shows the same fabrication processes illustrated in FIGS. **9a-9d**.

FIG. **10e** shows that the conductive metal **101** and **102** in FIG. **10d** is formed into signal traces **105** and **106**. The formation of the signal traces **105** and **106** can be achieved by using typical PCB fabrication techniques. Finally, the signal traces **105** and **106** are connected electrically with the conductive metal **107**.

FIG. **10f** shows that two sheets of dielectric material **108** and **109** are stuck to both sides of the substrate. To adhere the dielectric material to the substrate, the processes of heating and hardening will be needed.

FIG. **10g** shows that the top and bottom sides of the substrate are constructed with conductive metal **110** and **111**. Chemical vapor deposition or other known techniques can be used for constructing the conductive metal.

FIG. **10h** illustrates how the through hole **112** is constructed. Mechanical or laser drilling can be used for the construction of the through hole **112**. The gap (anti-pad) **114** between the through hole **112** and the plated via **114** can be properly designed to control the impedance of the signal via later.

FIG. **10i** shows that the through hole **112** in FIG. **10h**, which is plated with conductive metal **118**, is to be connected to the conductive metal **115** and **116** on the top and the bottom sides, respectively. However, the through hole **112** in FIG. **10h** can also be filled with conductive metal **123**, as shown in FIG. **10j**, to connect the conductive metal **120** and **121** on the top and the bottom sides, respectively.

Simulations were explored with printed circuit board (PCB) and integrated circuit packaging to confirm the performance of the two types of via structures that comprise the invention: type A (see FIG. **2**), a novel microstrip-like via, and type B (see FIG. **4**), a novel stripline-like via. FIG. **5** and FIG. **6** are the simulation results for the typical size of the PCB-level via structures. FIG. **5** shows the simulated insertion losses (S_{21}) of the microstrip-like via (see FIG. **1** and FIG. **2**). The simulation results for two via structures are shown: a) the via structure without power via (or ground via) (see FIG. **1**), and b) the via structure with power via (or ground via) (see FIG. **2**). The solid and dashed curves represent the via structures with and without the power via (or ground via), respectively. It is clearly shown that the insertion loss (S_{21}) of the via structure with the power via (or ground via) is larger than that of the via structure without the power via (or ground via) when the signal frequency is under about 1.07 GHz. These results indicate that the signal will be transmitted more effectively when the via structure with power via (or ground via) is used. In other words, the signal integrity is improved.

FIG. **6** shows the simulated insertion losses (S_{21}) of the stripline-like via (see FIG. **3** and FIG. **4**). Two via structures are simulated: a) the via structure without the power via (or ground via) (see FIG. **3**), and b) the via structure with the power via (or ground via) (see FIG. **4**). The solid and dashed curves represent the via structures with and without the power via (or ground via), respectively. Obviously, the insertion loss

(S_{21}) of the via structure with the power via (or ground via) is larger than that of the via structure without the power via (or ground via) when the signal frequency is under about 1.07 GHz. These results indicate that the signal will be transmitted more effectively when the via structure with the power via (or ground via) is used. In other words, the signal integrity is improved.

Similarly, FIG. 7 and FIG. 8 are the simulation results for the IC packaging-level via structures. FIG. 7 shows the simulated insertion losses (S_{21}) of the microstrip-like via (see FIG. 1 and FIG. 2). The solid and dashed curves represent the via structures with and without the power via (or ground via), respectively. Obviously, the insertion loss (S_{21}) of the via structure with the power via (or ground via) is larger than that of the via structure without the power via (or ground via) when the signal frequency is under 20 GHz. These results indicate that the signal will be transmitted more effectively when the via structure with the power via (or ground via) is used. Consequently, the signal integrity will be improved.

FIG. 8 shows the simulated insertion losses (S_{21}) of the stripline-like via (see FIG. 3 and FIG. 4). The solid and dashed curves represent the via structures with and without the power via (or ground via), respectively. Obviously, the insertion loss (S_{21}) of the via structure with the power via (or ground via) is larger than that of the via structure without the power via (or ground via) when the signal frequency is under 20 GHz. These results indicate that the signal will be transmitted more effectively when the via structure with the power via (or ground via) is used. Consequently, the signal integrity will be improved.

The embodiments of the invention comprise the improved via structures (the additional power via or ground via), both via types having the same method of fabrication. The power via (or ground via) will connect the two power planes (or the two ground planes). It provides the shortest current return path and reduces the impedance discontinuity. The thickness of the power via (or ground via) must be larger than the skin depth associated with the frequency of the signal. Therefore, the EM waves generated by the signal via will not penetrate through the power via (or ground via). Therefore, the insertion loss will be improved. Also, the EM field will be concentrated between the signal via and power via (or ground via) and, consequently, the signal via will couple less energy with any other via. As a result, the signal integrity will be improved.

In short, the inventive concepts unveiled herein are the basic examples and can be modified. Any modification based on any extension of the inventive concepts fall within the scope of the appended claims and their equivalents; and consequently, they should be covered by this patent.

What is claimed is:

1. A via structure in a circuit board or substrate having multiple layers, from top to bottom comprising:

- a first power plane or a first ground plane in the first layer;
- a first medium in a second layer;
- a first signal trace in a third layer;
- a second medium in a fourth layer;
- a second signal trace in a fifth layer;
- a third medium in a sixth layer;
- a second power plane or a second ground plane in a seventh layer;

a power via or a ground via passing through the first medium, the first signal trace, the second medium, the second signal trace, and the third medium, wherein the first power plane or the first ground plane is connected

with the second power plane or the second ground plane through the power via or the ground via;

a signal via passing through the second medium around the power via or the ground via, and the first signal trace is connected with the second signal trace through the signal via; and

an anti-pad is located between the signal via and the power via or a ground via,

wherein the power via or the ground via is cylindrical and the thickness of the power via or the ground via is larger than the skin depth at the frequency of the signal; and when a signal going through a path from the first signal trace through the signal via to the second signal trace, the signal goes through the path with complete impedance continuity.

2. The via structure of claim 1, wherein the first power plane or the first ground plane in the first layer and the second power plane or the second ground plane in the seventh layer have to have the same potential.

3. The via structure of claim 1, wherein the power via or the ground via is either a solid cylindrical or a cylindrical ring.

4. The via structure of claim 1, wherein the signal via are concentric to the power via or the ground via.

5. The via structure of claim 1, wherein the circuit structure comprises a printed circuit board and integrated circuit packaging.

6. The via structure of claim 1, wherein the power plane or the ground plane is deformed to a power grid or a ground grid on die.

7. A via structure, comprising:

a circuit board having multiple power planes, ground planes and medium layers, wherein a power via or a ground via passes through the power planes, ground planes and medium layers along a same axis, and wherein a signal via passes through the medium layers around a power or the ground via;

one of the multiple power planes or the ground planes connects with a different power plane or the ground [via] plane through the power via or the ground via;

the power via or the ground via may connect all of the multiple power planes or the ground planes with same potential;

the power via or the ground via is cylindrical and the thickness of the power via or the ground via is larger than the skin depth at the frequency of the signal; and

when a signal going through the signal via, the signal goes through the signal via with complete impedance continuity.

8. The via structure of claim 7, wherein the signal via is concentric with the power via or the ground via.

9. The via structure of claim 7, wherein the signal via and power via or the ground via can be not only circular but also square, rectangular, or any shape.

10. The via structure of claim 7, wherein the distance between the signal via and the power via or the ground via is properly designed for impedance control of the signal via.

11. The via structure of claim 7, the dielectric material between signal via and the power via or the ground via is properly selected for impedance control of the signal via.

12. The via structure of claim 7, wherein the signal via and the power via or the ground via are perpendicular to the power planes or the ground planes of the circuit board.

13. The via structure of claim 7, wherein the signal via and power via or the ground via are located in the dielectric.