NAVMAN

Jupiter 32

Miniature 20-channel GPS receiver module

Integrator's Manual

Related documents

- Jupiter 32 Product Brief LA000268

- Jupiter 32 Data Sheet LA000267

- Jupiter Development Kit Guide LA000645

- Low Power Operating Modes Application Note LA000513

- Jupiter 32/30 Saving and retrieving configuration data to Flash and Selectable user profiles Application Note LA000266

- Navman NMEA Reference Manual MN000315

- SiRF Binary Protocol Reference Manual, MN000314

SiRF Powered

Contents

Figures

Tables

1.0 Introduction

The Navman Jupiter 32 module is a complete GPS receiver designed for surface mount assembly integration. The Jupiter 32 provides a simple, cost effective GPS solution for application designers. Application integration will vary primarily with respect to antenna system design and EMI protective circuitry.

The Jupiter 32 is the successor to the established Jupiter 30, being electrically compatible and having a very small form factor. The provides an easy migration path for existing users requiring very small packaging, low cost, high volume, greater sensitivity, lower power consumption and a faster fix.

Basic operation requires a power supply, GPS antenna system interface, relevant EMI protection, and the design and layout of a custom PCB. This document outlines the following design considerations and provides recommended solutions:

Hardware application information
This section introduces the system interface and provides the following physical specifications:

    a. electrical connections (SMT pad interface)

    b. mounting (PCB pad layout dimensions)

    c. manufacturing recommendations

    d. application circuit interface

It also discusses fundamental considerations when designing for RF, and presents the antenna system design overview. This covers the following topics:

    a. PCB layout

    b. antenna system design choices

A sample solution is presented and discussed. Due to the nature and complexity of GPS signals, it is recommended that application integrators adhere to the design considerations and criteria described in this document.

Software application information
This section provides answers to some common questions that might not have been covered in the above topics.

2.0 Hardware application information

The Jupiter 32 module provide 34 Land Grid Array (LGA) pads for electrical connections. The sections that follow introduce the physical and relative functional specifications for application integration.

Note: The electrical connections can carry very low level GPS signals at 1.57542 GHz. The layout must be designed appropriately with consideration of the frequencies involved.

2.1 Electrical connections (SMT pad interface)

The round hole on the front label side of the chip locates pad A1. The pads are designated A-F and 1-7. Details of the pad layout and numbering are shown in Figure 2-1.

Pad No	Pad Names	Type	Description
A1	RF_IN	I	RF Input 50 ohm
A2	GND	P	RF signal ground return
A3	GND	P	ground
A4	VANT	P	active antenna power input
A5	GPIO15	I/O	reserved
A6	GPIO14	I/O	reserved
A7	RF_ON	O	output to indicate whether the RF section is enabled (active high)
B1	GND	P	RF signal ground return
B3	VDD_BB	P	baseband power supply input for external regulator build option - DO NOT CONNECT
B4	GPIO4	I/O	reserved
B5	GPIO13	I/O	reserved
B6	VDD_RTC	P	RTC power supply input for external regulator build option - DO NOT CONNECT
B7	GPIO1	I/O	reserved
C1	GND	P	ground
C2	TCXO_IN	I	TCXO input for external TCXO build option - DO NOT CONNECT
C4	BOOT	I	active high to download flash firmware
C5	LNA _EN	O	external antenna enable, active high
D2	FACTORY USE ONLY		reserved
D4	N_GPS_FIX	O	active low when 2D or 3D fix data available
D5	RXA	I	CMOS level asynchronous input for UART A
D7	TXA	O	CMOS level asynchronous output for UART A
E1	GND	P	ground
E2	GND	P	ground
E4	GPIO2	I/O	reserved
E5	N_RESET	I	master reset, active low
E6	RXB	I	CMOS level asynchronous input for UART B
E7	1PPS	O	1 pulse per second output 1µs wide
F1	PWRIN	P	3.0 - 3.6 V main power supply input
F2	VBATT	P	1.9 - 3.6 V backup battery input
F3	VCC_RF	P	RF power supply output for external regulator option. Connect as per application circuit. - Figure 2-2
F4	WAKEUP	I	Push-to-Fix wake-up, active high
F5	N_WAKEUP	O	low output indicates baseband is powered up - DO NOT CONNECT
F6	TXB	O	CMOS level asynchronous output for UART B
F7	ECLK	I	external clock input, not used

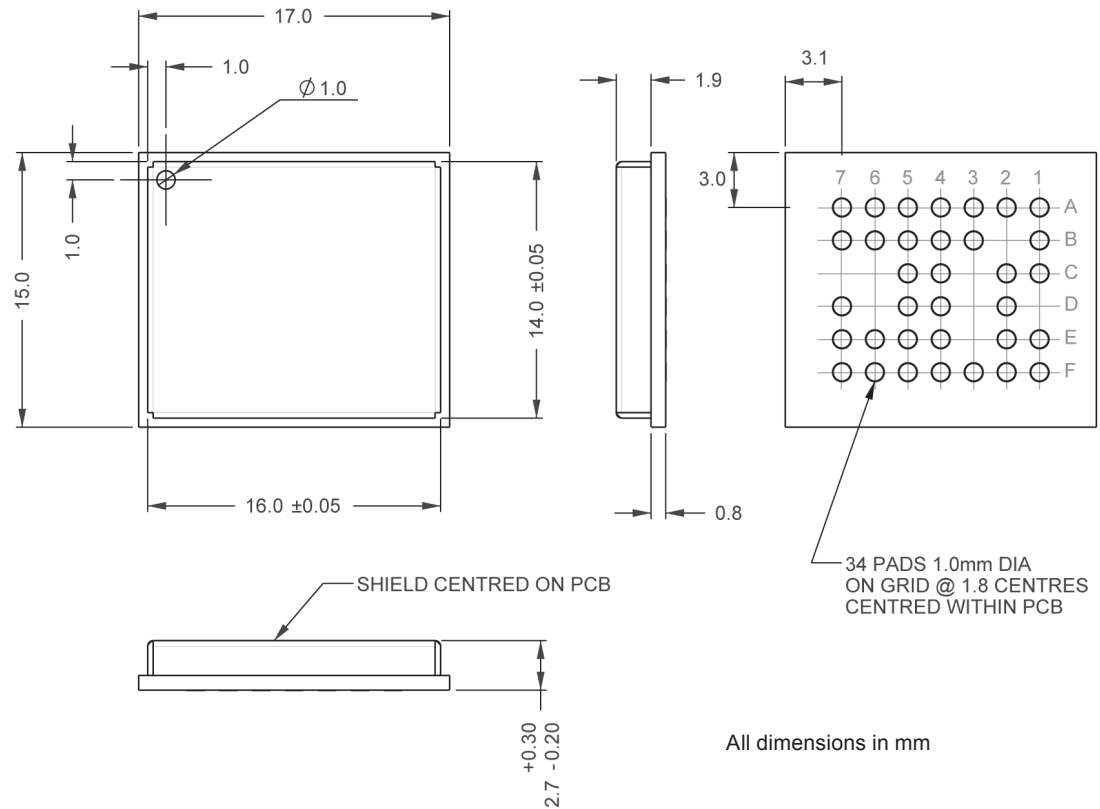**Table 2-1: Jupiter 32 Module pad functions**

Figure 2-1: Jupiter 32 mechanical layout

2.2 Typical application circuit

The schematic in Figure 2-2 (next page) represents a very basic application circuit, with simple interfaces to the module. It is subject to variations depending on application requirements.

2.2.1 Power for receiver and active antenna

The receiver power connection requires a clean 3.3 VDC. Noise on this line may affect the performance of the GPS receiver.

When an active antenna is used, the DC power is fed to it through the antenna coax. This requires the user to apply the antenna DC voltage to pad A4 of the module.

A 2.85 V 30 mA supply is made available on pad F3 if the chosen antenna can accept that voltage. This supply is under the command of the TricklePower energy control.

The source impedance of the power supply must be kept sufficiently low. The capacitance of the power supply and PCB track width determine the overall source impedance. If a passive antenna is used, the supply must be able to provide a minimum of 100mA continuously with minimal ripple as measured at the power input pin. This ripple requirement must be adhered to for a larger current flow when an active antenna is used. In-rush current for some active antennas have been measured as high as 70mA and may cause a dip in voltage or ripple. Therefore, a low source impedance is important since the power to the Jupiter 32 must be able to accommodate instantaneous currents in excess of 200mA.

2.2.2 Grounding

Separate analogue and digital grounds are not used. However, the antenna signal ground uses a particular layout for optimum results. See Figure 2-3 (next page) and Section 2.3.5 for ground plane recommendations and for design considerations involving the antenna input and the 50 ohm microstrip connection.
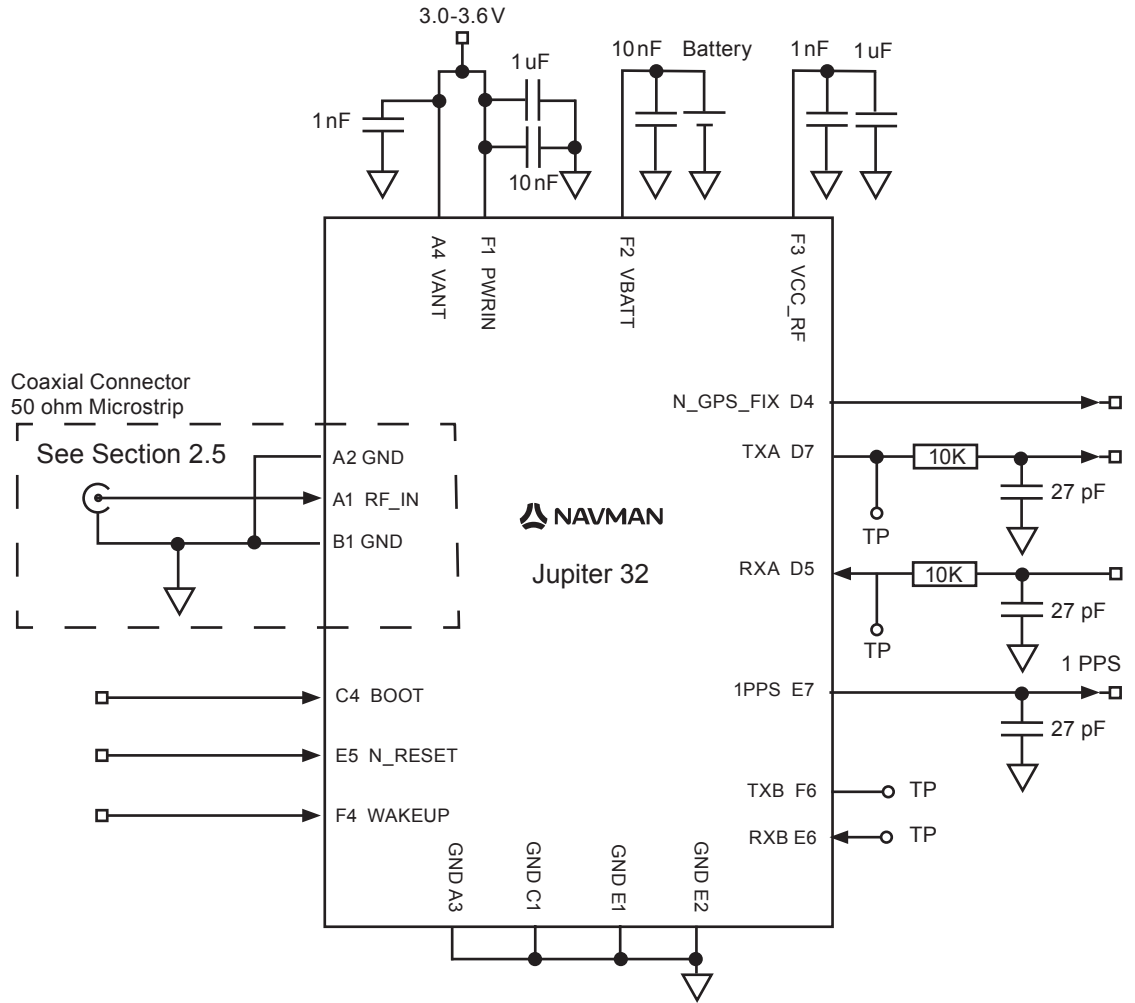
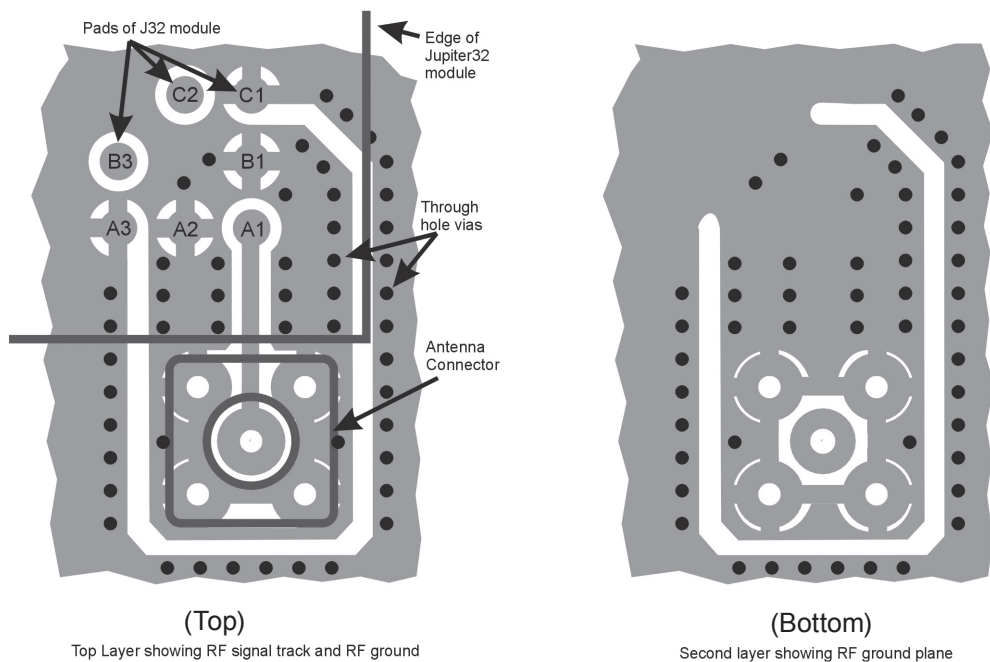**Figure 2-2: Basic Jupiter 32 application circuit**

(Top)
Top Layer showing RF signal track and RF ground

(Bottom)
Second layer showing RF ground plane

Figure 2-3: Example PCB layout for external active antenna

2.2.3 Decoupling

The schematic in Figure 2-2 illustrates a suggested method of decoupling that may be followed. Table 2-2 suggests decoupling values for all signals relative to the function required. This level of decoupling may not be required in a particular application, in which case these capacitors could be omitted. As shown in Figure 2-2, only the signal lines used in the application require decoupling.

All capacitors are highly recommended if the module will experience substantial electromagnetic interference (EMI). All low value capacitors should be as close as possible to the module pad with a short connection to the ground plane (see Figure 2-4). Any data lines that have not been properly shielded are susceptible to data corruption.
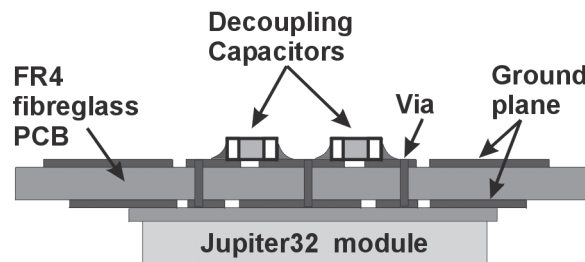

Figure 2-4: Decoupling Capacitor Placement

2.2.4 Serial RS232 data level shifter

To connect the module to a PC COM port, the serial data signals must be level shifted to RS232 levels. This has not been shown in the reference design, but many single chip RS232 level shifters are available, such as MAX3232.

Note: It is highly recommended to provide test points on the serial data lines and 'BOOT' signal (pad C4), even if the application circuit does not use these signals. This will allow the user to connect to these signals if a firmware upload or new configuration is required. These test points can take the form of an unfitted 'through-hole' connector. Refer to Figure 2-2.

2.3 PCB design recommendations

The modules are surface mounted devices, hence the layout of the application PCB plays an integral part in the overall performance of the finished system.

It is not difficult to design such a PCB, despite the presence of high frequency, low level radio signals. The following recommendations have been offered to allow the designer to create a design that will meet the requirements of this product.

Figure 2-1 shows the copper pad dimensions and layout.

2.3.1 Choice of PCB stack up

In general, a two layer PCB substrate can be used, with all the RF signals on one side. Multi-layer boards can also be used. The design recommendations here only address the microstrip style of RF connection. Stripline designs can also be accommodated, however, long lengths of stripline can cause excessive signal loss and vias in the signal track should be avoided.

2.3.2 Ground plane design

A complete ground plane should be used under the PCB with signal tracks on the same layer as the module. The ground return for any signal should ideally have a clear path back to its source and should not mix with other signal's ground return paths. For this reason, RF signal ground, underneath the microstrip antenna connection, should not be shared with any digital signal or power supply return paths. Pads A2 and B1 are the RF signal ground return connections.

2.3.3 Decoupling components

The recommended values for power and signal decoupling are shown in Table 2-2. The placement of these components must ensure that the low value capacitors have very short connections to the module pad and to the ground plane.

Function	Pad	Decoupling
PWRIN	F1	1 µF∥10 nF
BOOT	C4	27 pF
RXA	D5	27 pF
TXA	D7	27 pF
TXB	F6	27 pF
RXB	E6	27 pF
RF_ON	A7	27 pF
VANT	A4	1 nF
VCC_RF	F3	1 µF∥1 nF
VBATT	F2	10 nF
N_RESET	E5	27 pF
N_GPS_FIX	D4	27 pF
GPIO (ALL)	-	27 pF
WAKEUP	F4	27 pF
LNA_EN	C5	27pF
1PPS	E7	27 pF
Note: '∥' represents a parallel connection		

Table 2-2: Decoupling recommendations

2.3.4 Antenna connection

The PCB layout design of the antenna input connection requires the appropriate selection of PCB track width, substrate material and careful attention to the layout geometry. If this overall system is not implemented correctly, the module will receive poor GPS signals and therefore provide inferior navigation data.

Figure 2-5 shows an example of a PCB design integrating an external active antenna connected via an MCX or SMA coaxial connector.
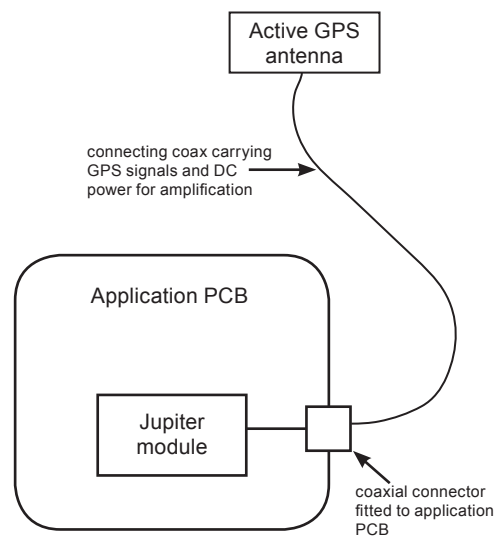

Figure 2-5: Arrangement of active antenna and application board

The modules can be used with a passive patch antenna if the connection to the antenna input is very short. It is possible to mount the patch antenna on the same PCB as the module, but to reduce the possibility of digital noise, it is recommended that the antenna be mounted on the opposite side of the board to the module. (Figure 2-6 shows an example of a PCB design integrating a passive patch antenna.)
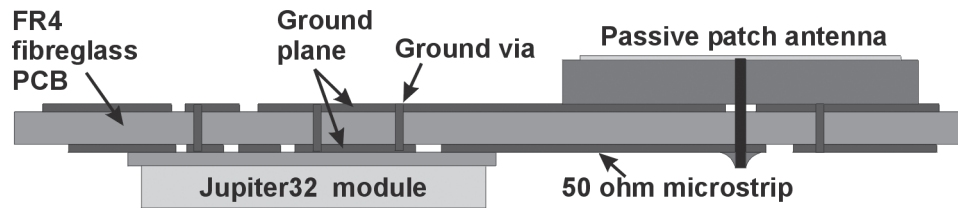
Figure 2-6: Cross section of application board with passive patch antenna

2.3.5 Design of 50 ohm microstrip antenna connection

When designing the signal track from the antenna connection to the antenna input on the module, a controlled impedance microstrip with a characteristic impedance of 50 ohms at 1.5 GHz must be used.

The PCB parameters that affect impedance are as follows:
1. Track width (W)
2. PCB substrate thickness (H)
3. PCB substrate permittivity (ε_r)
4. To a lesser extent, PCB copper thickness (T) and proximity of same layer ground plane.

Figure 2-7 shows a representation of the PCB microstrip and its parameters.
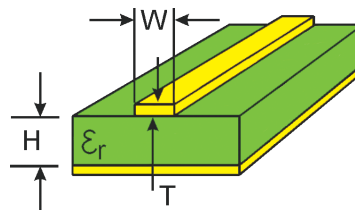
Figure 2-7: PCB microstrip dimensions

Table 2-3 shows typical track widths for an FR4 material PCB substrate (permittivity ε_r of 4.3 at 1.5 GHz) and different PCB thickness. The effect of track thickness (T) can be ignored for the short track lengths associated with this design.

Substrate material	Permittivity ε_r	Substrate thickness H (mm)	Track width W (mm)
FR4	4.3	1.6	2.0
		1.2	1.8
		1.0	1.6
		0.8	1.4
		0.6	1.2
		0.4	0.7
		0.2	0.4

Table 2-3: PCB substrate thicknesses v track width

Notes:
1. If a multi-layer PCB is used, the thickness is the distance from signal track to nearest ground plane.
2. If the antenna connection is routed under the module, the track width should be approximately halved for that section only.

It is recommended that the antenna connection PCB track should be routed around the outside of the module outline, kept on a single layer and have no bends greater than 45 degrees. It is not recommended (for production reasons) to route track under the module, but if track has to route under the module, it should have a modified track width and solder mask to avoid short circuits to the underside of the module.

To minimise signal loss and reduce the requirement for vias, it is recommended the signal track not be placed on an inner layer of a multi-layer PCB.

The PCB track connection to the RF antenna input **must**:
 • have a characteristic impedance of 50 ohm
 • be as short as possible
 • be interfaced to a coaxial connector if an external antenna is used
 • have max clearance to ground on the same layer, or at least be half the substrate thickness
 • be routed away from noise sources such as: switching power supplies, digital signals, oscillators and transmitters

The PCB track connection to the RF antenna input must **not** have:
 • vias
 • sharp bends
 • components overlaying the track

2.4 Antenna system design choices

2.4.1 Antenna types

The role of the antenna is to filter, amplify and down-convert the incoming GPS signals into an electric signal that can be processed by the receiver electronics within the RF section. There are several designs of GPS antennas:

 Monopole, or dipole, configurations.

 Quadrifilar helices.

 Spiral helices.

 Microstrips - active and passive.

 Planar rings ("choke ring"), and other multipath-resistant designs.

There are special considerations for GPS antennas. They must be able to pick up and discriminate very weak signals. GPS signals are circularly polarised, so the GPS antenna must also be circularly polarised. The antenna gain pattern design is intended to enhance the ability of the RF section to filter multipath and low elevation signals. An essential requirement of any consumer GPS product is that the antenna is a stable electrical centre which is coincident with the geometric centre and insensitive to the rotation and inclination of the antenna.

The main parts of the construction of the antenna consists of: (a) the omnidirectional antenna element, (b) the antenna preamplifier electronics, and (c) a ground plane (though not always present). The industry has widely adopted the patch antenna as the most common and practical design.

There are two major types of GPS antenna patch antennas: passive and active. The active antenna has a built in LNA (Low Noise Amplifier) to increase the strength of the signal, and to compensate for the signal loss in a long cable connection.

The features of each type of antenna are shown in Table 2-4, comparing an externally mounted active antenna with a passive patch antenna mounted on the same PCB as the module.

Feature	Passive antenna	Active antenna
antenna requires close proximity to receiver	yes	no
consumes power	no	yes
can be mounted remote from receiver	no	yes
gives good performance in poor signal situations	no	yes
has built in additional filtering	no	yes
low cost	yes	no
requires a coaxial connector	no	yes

Table 2-4: Passive and active Patch antenna features

2.4.2 Active antenna

An active antenna comprises a passive antenna with a built in LNA that requires a power supply. Active antennas are used when the antenna input is connected to the receiver through a coaxial cable (usually longer than 3 m) or any high loss transmission path. The GPS signals experience loss in the transmission path from the antenna. The loss is overcome by the antenna's LNA, which amplifies the signal before it enters the transmission path.

The amplification is also used to enhance the signal in areas of low signal. If the coaxial cable is shorter than 3 m it may experience too much gain at the receiver and degrade the performance. There are some variations as to how the antenna will receive its power, but it is usually supplied through the coaxial cable via the antenna input as shown in Figure 2-5.

Refer to Table 2-5 for the recommended active antenna characteristics.

2.4.3 Passive antenna

A passive antenna does not require any power because it has no amplifier. This is not the best choice if signal strength is a concern, however, it may be sufficient if the signal path is kept to a minimum (usually below 300 mm). An advantage to using a passive antenna is the ability to mount directly onto the application. For best performance, a passive patch antenna should have a metal ground plane (about 80 mm in diameter) placed directly under the antenna, and it is advisable to shield the module and application circuits from the antenna. For this reason the antenna and Jupiter 32 module should not be mounted on the same side of the PCB (see Figure 2-6).

Any cover close to the antenna (called the superstrate) will cause the resonant frequency and efficiency of the antenna to drop. It is therefore recommended to keep any distance to the superstrate to a minimum of 3 mm from the top surface of the patch.

See Table 2-5 for recommended characteristics of both passive patch and active antennas for use with the Jupiter receiver.

Characteristic	Active antenna	Passive antenna
polarisation	right-hand circular polarised	right-hand circular polarised
receive frequency L1	1.57542 GHz +/- 1.023 MHz	1.57542 GHz > +/- 1.023 MHz
power supply	3 V (typ), 5 V max.	–
DC current	< 10mA at 3 VDC	–
antenna gain	–	+2 to 5 dBi with 1 dB loss (max) in connections
total gain (includes antenna gain, LNA gain and cable loss)	≤ 26 dBi (Jupiter 20) ≤18 dBi (Jupiter 32)	–
axial ratio	< 3 dB	< 3 dB
output VSWR	< 2.5	–

Table 2-5: Recommended antenna characteristics

Note: GPS active and passive antenna selection must include practical TTFF tests in weak and strong outdoor environments – noting peak and average signal strength measurements. This must be done in comparison with the antenna supplied in the Development kit. Performance results and signal strength measurements must be comparable to the reference antenna supplied. Passive antenna signal strength measurements will be on average lower than an active antenna. This depends on size of patch antenna/ground plane, distance from RF input, gain of the element and noise in the application circuit. However, the TTFF should be comparable in open sky conditions. An active antenna is recommended if peak performance is required in very weak or obscured conditions.

2.4.4 Jupiter module used as a GPS sensor

The adapter board reference design shows how a Jupiter 32 GPS receiver module can be used with an external active antenna via a coax connector (See Figure 2-5). The same design can be used with a passive patch antenna on the same PCB. The module is placed so the connection between the antenna and the antenna input pad is as short as possible. Also note, the PCB should have a complete ground plane on the patch side of the board which serves as the ground plane required by the antenna (See Figure 2-6).

The serial data from the module must be connected to a local host processor, and care should be taken so noise from these devices cannot enter the signal path or GPS antenna. It is recommended that all digital devices are placed on the opposite side of the board from the antenna.

2.4.5 DC supply protection for an active antenna

Antenna DC supply current limit
When the Jupiter 32 receiver is used with an external active antenna, the DC supply in the coax cable is vulnerable to over-current if a fault occurs in the antenna. For example, this can happen if its cable gets crushed in a car door.

Warning: The Jupiter 32 module antenna power feed does not have internal current limiting. Damage can occur if unlimited current is permitted to flow through the module antenna power feed components.

The circuit shown in Figure 2-8 (next page) will provide over-current protection.
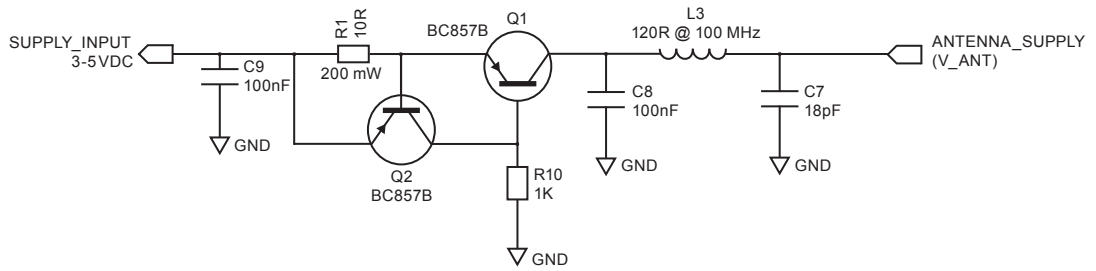
70 mA Antenna supply current limit

Figure 2-8: Simple current limiter circuit

NOTE: Ensure that the In-rush current of your active antenna does not cause it to approach the current limit.

Transistor Q1 serves as a series pass transistor. Q2 is used to sense the current in the antenna circuit, turning off Q1 if the voltage across the current sense resistor R1 exceeds 0.6 V. This circuit does not turn off the supply to the antenna, but merely limits it to a safe value. With the components shown the supply will be limited to approximately 70 mA.

Other options are available to provide this function. MAXIM can supply a current trip IC that will turn off the load if a preset supply current is exceeded. The part number for this is MAX4785. The circuit is shown in Figure 2-9.
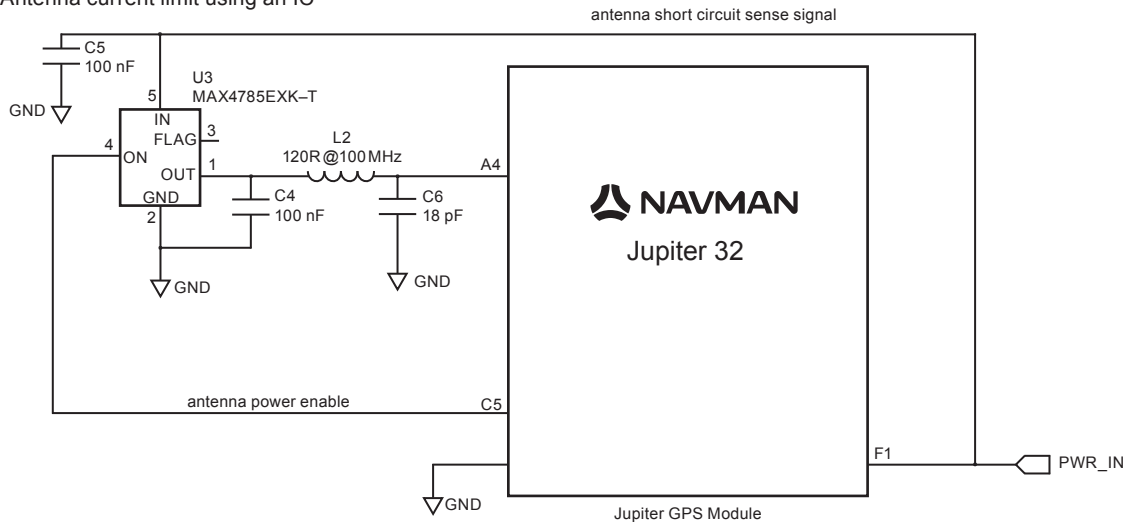

Antenna current limit using an IC

Figure 2-9: External Active Antenna Control and Current Limit circuit

Electro-resistive fuses can be used to protect the supply over-current. These are available from Bourns, Vitromon and other manufacturers.

2.5 Jupiter adapter printed circuit board

The Jupiter 32 module supplied in the Development Kit is mounted on a carrier PCB in a method typical of a customer application. This carrier PCB illustrates and implements many of the design considerations discussed in this integrator's manual.

The module is interfaced through a downward-facing 20-pin data connector.

The RTC (Real-Time Clock) backup supply voltage can be provided by an on-board lithium cell, or from the Development unit backup supply. The adapter board carries a switch debounce IC to provide reliable reset action from the Development unit.

Figure 2-10 (next page) shows the adapter board with the positions of the connectors and indicators.
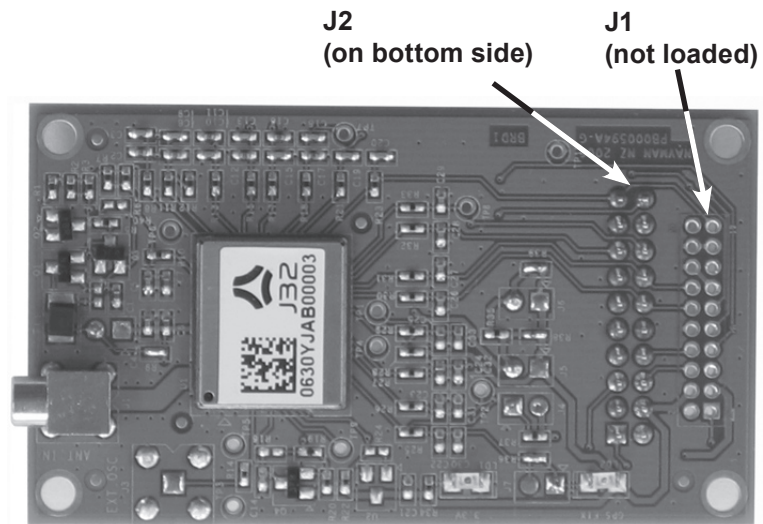
**J2
(on bottom side)**　　　**J1
(not loaded)**

Figure 2-10: Jupiter 32 adapter board

Refer to Table 2-6 for a description of the connector interfaces.

Jupiter function	J2 (2.54 mm pitch header) pin no.
VANT	1
no connection	2
VBATT	3
VDD	4
N_RESET	5
reserved	6
reserved	7
BOOT	8
reserved	9
RF_ON	10
TXA	11
RXA	12
reserved	13
TXB	14
RXB	15
WAKEUP	16
GND	17
reserved	18
1PPS	19
N_GPS_FIX	20

Table 2-6: Connector configuration

3.0 Packaging and delivery

Jupiter 32 modules are shipped in Tape and Reel form. The reeled modules are shipped with 250 units per reel. Each reel is 'dry' packaged and vacuum sealed in an Moisture Barrier Bag (MBB) with two silica gel packs and placed in a carton.

All packaging is ESD protective lined. The Jupiter 32 GPS receiver is a Moisture Sensitive Device (MSD) level 3. Please follow the MSD and ESD handling instructions on the labels of the MBB and exterior carton. See Figures 3-1, 3-2 and 3-3.

CAUTION

This bag contains
MOISTURE SENSITIVE DEVICES

1. Calculated shelf life in sealed bag: 12 months at < 40°C and 90% relative humidity (RH).

2. Peak package body temperature: _____240_____ °C,
   If blank see adjacent bar code label

3. After bag is opened, devices that will be subject to reflow solder or other high temperature process must be:
   a) mounted within _____48_____ hours, at factory conditions
   If blank see adjacent bar code label
   ≤ 30°C/60% RH, or
   b) stored at ≤ 10% RH.

4. Devices require baking before mounting if:
   a) humidity indicator card is > 10% when read at 23±5°C,
   b) condition 3a or 3b not met.

5. If baking is required, devices may be baked for 48 hours at 125±5°C.

Note: If device containers cannot be subjected to high temperature or shorter bake times are desired,

reference: IPC/JEDEC J-STD-033 for bake procedure.

Bag seal date _____ °C,
If blank see adjacent bar code label

Note: Level and body temperature defined by IPC/JEDEC J-STD-020

Figure 3-1: Moisture Barrier Bag Label

330.0

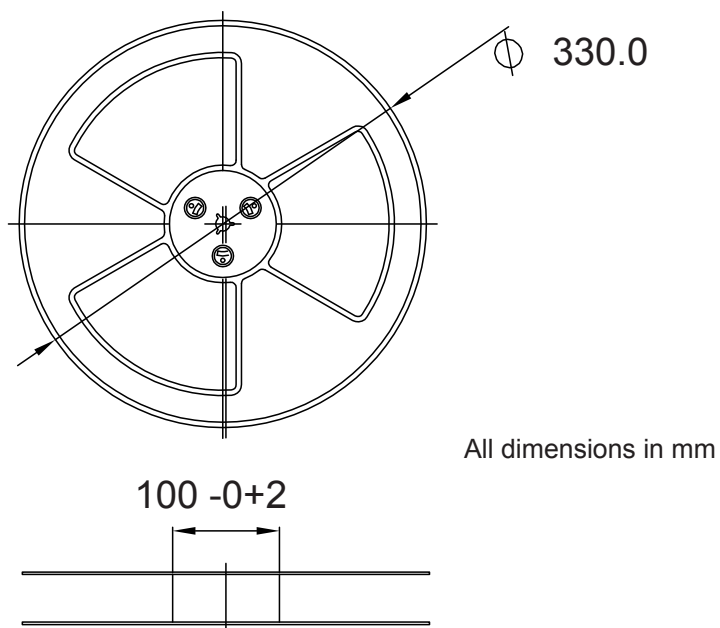All dimensions in mm

100 -0+2

Figure 3-2: Reel dimensions

250 per reel

44mm

20.00

2.00

1.5

4.00

1.75

20.20

W 44

2.0

Direction of Feed

3.2±0.10

0.35±0.05

13.30±0.10
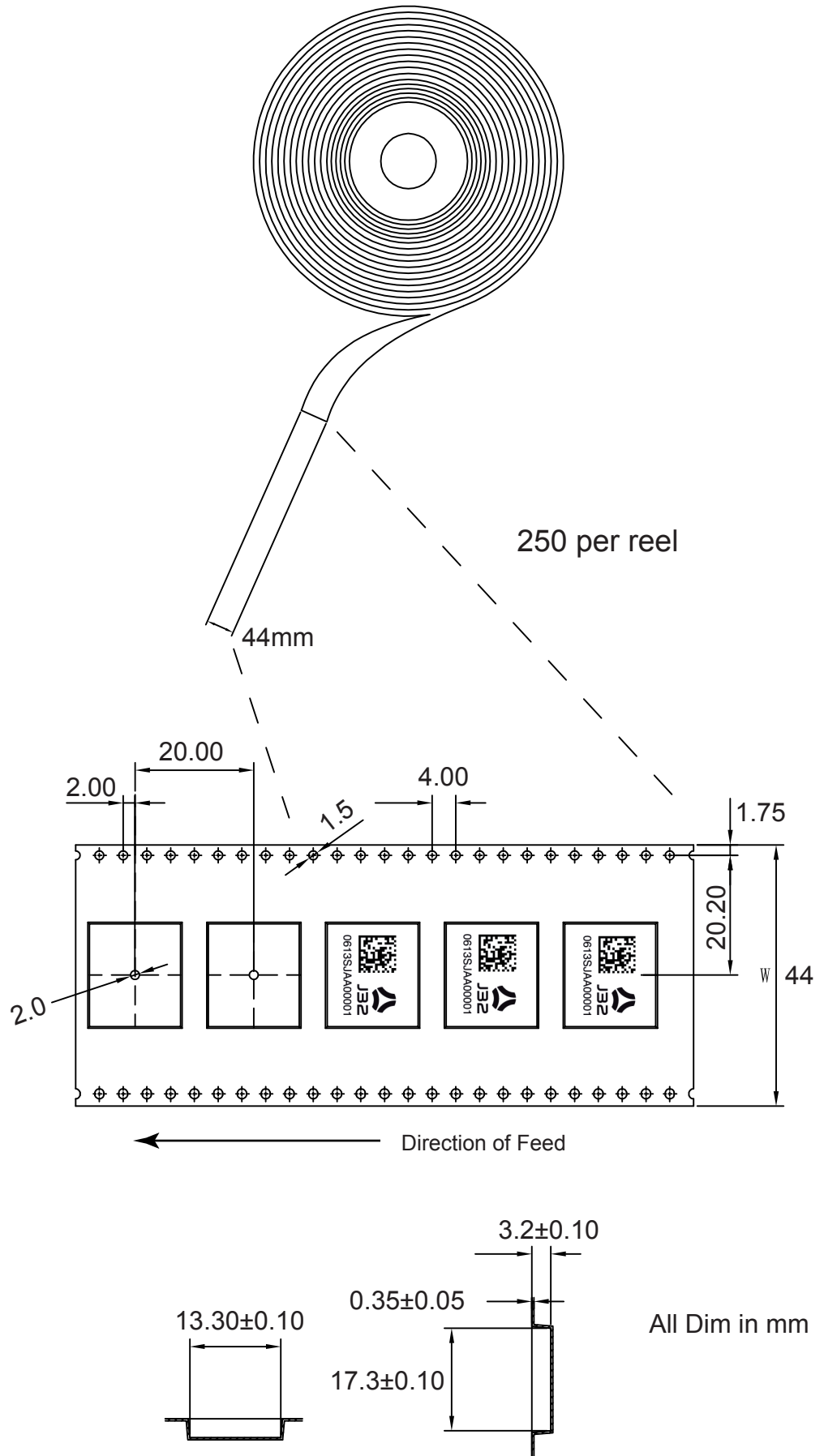
17.3±0.10

All Dim in mm

Figure 3-3: Jupiter 32 Packaging

4.0 Manufacturing process recommendations

The Jupiter 32 uses the latest Land Grid Array (LGA) technology. Solder interconnect is formed solely by solder paste applied to the board assembly. This results in a low stand-off height, depending on solder paste volume and Printed Circuit Board (PCB) geometry. This makes LGA ideal for small form-factor applications. Solder joint reliability studies indicate that LGA greatly exceed typical industry reliability.

4.1 Solder methods

The Jupiter 32 has 34 surface mount connection pads that have a base metal of copper with an Electroless Nickel Immersion Gold (ENIG) finish. This is suitable for a lead free manufacturing process. Critical factors to ensure successful circuit board assembly with LGA devices include, the design of the solder paste stencil, the solder paste and reflow profile used. Stencil attributes that have been shown to succeed in tests are solder stencil thickness, aperture diameter, paste release characteristics, and practices to ensure consistent solder paste volumes that exceed recommended minimums.

4.1.1 Solder paste type

The module will accept all commonly used solder pastes. The solder paste can be lead based or lead-free. In all processes, factors such as circuit board thickness, fabrication complexity, assembly process compatibility, and surface finish should be taken into consideration. Also, due to the low standoff height of LGA devices and difficulty cleaning, a no-clean paste is recommended.

4.1.2 Solder paste mask size

Solder paste mask size should be adjusted by experimentation according to the production process requirements. As a starting point, a 0.75:1 (paste mask:pad size) ratio is recommended.

4.1.3 Solder paste stencils

A 0.125 mm (5 mil) thick stencil is recommended with LGA.

4.1.4 Reflow profile recommendations

An optimal reflow profile depends on solder paste properties and should be optimised and proven out as part of an overall process development. The following guidelines represent good soldering practices to help yield high quality assemblies with minimum rework.

It is important to provide a solder reflow profile that matches the solder paste supplier's recommendations. Some fluxes need a long dwell time below the temperature of 180°C, while others will be burned up in a long dwell. Temperatures out of bounds of the solder paste flux recommendation could result in poor solderability of components on the board. Solder paste suppliers can recommend an suitable reflow profile to give the best solderability. The maximum reflow temperature is 260°C for 10 seconds.

The typical reflow profile consists of four sections. In the preheat section, the PCB assembly should be preheated at the maximum rate of 1° to 2°C/sec to start the solvent evaporation and to avoid thermal shock. The assembly can then be heated for 60 to 120 seconds to remove solder volatiles and activate the flux so the temperature of the package rises above 100°C. The reflow section of the profile, the time above liquidus (approx. 185°C), can be between 45 to 60 seconds with a peak temperature in the range of 230° to 260°C. This allows all solder joints on the assembly to fully reflow and a maximum peak temperature that is below the maximum that the devices or solder material can tolerate. Finally, the assembly can undergo cool-down in the fourth section of the liquidus point (approx. 185°C). The actual profile parameters depends upon the recommendations of the solder paste supplier and other reflow requirements of the customer's PCB.

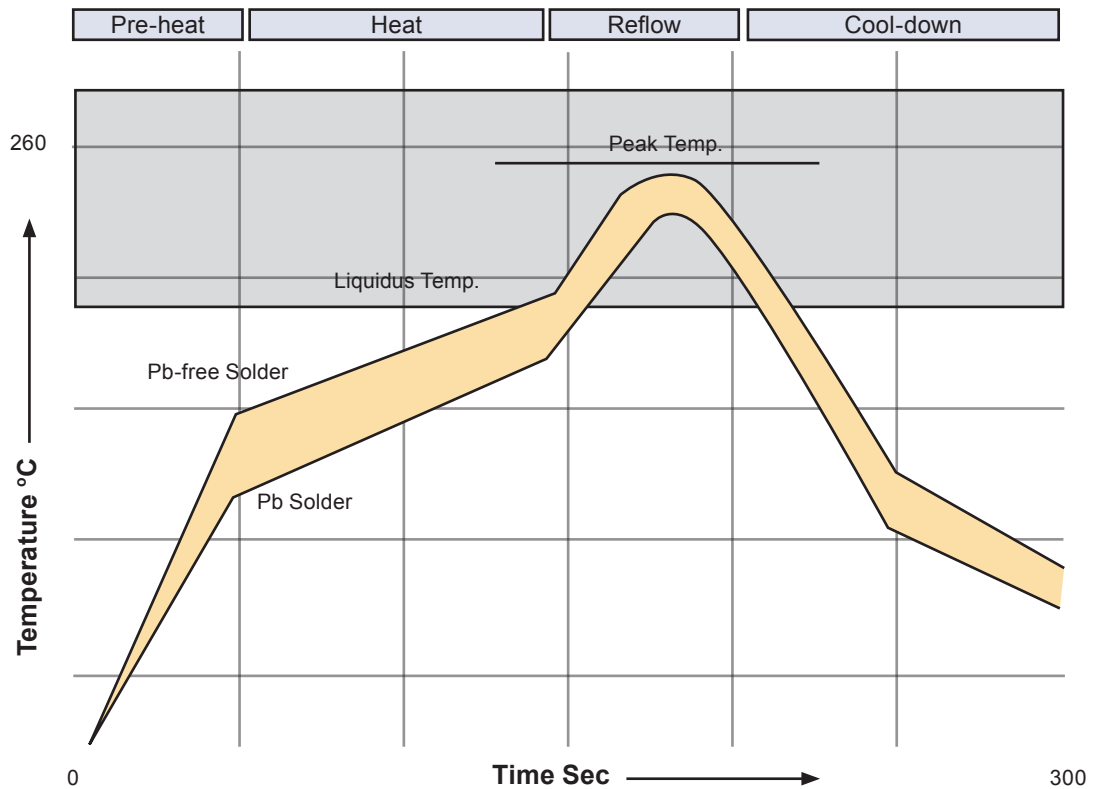Refer to Figure 4-1 for sample lead and lead-free reflow profiles.

Figure 4-1: Sample Lead and Lead free reflow profile

4.1.5 Coating

The final PCB may be selectively coated with an acrylic resin, air / oven cured conformal coating, clear lacquer or corresponding method, which gives electrical insulation and sufficient resistance to corrosion.

4.1.6 Post reflow washing

It is recommended that a low residue solder paste is used to prevent the need for post reflow washing. If a washing process is used, an aqueous wash is not recommended due to the long drying time required and danger of contaminating the fine pitch internal components.

4.1.7 Pre-baking

The modules will be delivered on a tape and reel package, and sealed in an airtight bag. The MSR (Moisture Sensitivity Rating) is 3, therefore they should be loaded and reflowed within 168 hours (7 days). If the modules are in ambient humidity for longer than this, a pre-baking/ drying process will be required.

4.1.8 Rework

It is recommended that no more than 2 reflow cycles are performed and that the maximum reflow temperature recommended by the solder paste manufacturer is not exceeded. The maximum ramp-up rate of 3° C/sec for leaded solder or 4° C/sec for lead free solder should not be exceeded.

Navman recommends that rework and repair is carried out in accordance with the following guidelines:

• IPC-7711 Rework of Electronic Assemblies

• IPC-7721 Repair and Modification of Printed Boards and Electronic Assemblies

For proper removal of the part, special nozzles should be used for local heating of the part to avoid reflow of adjacent parts. The PCB assembly should be preheated to about 75° C from the bottom using convective pre-heaters and the preheated nozzle should be lowered

on the part. The proper nozzle should also heat the component leads by either hot gas or hot bar. The ideal reflow profile should be the same as the one used for mounting the part and depends upon the paste used. The reflow zone can be shortened as long as the reflow is complete. The part should then be lifted off automatically during the transition from reflow to cool down cycles using a vacuum.

4.1.9 Site Redress and Cleaning

Once the part is removed, the site needs to be cleaned for attachment of a new package. This may be done by vacuum de-soldering or wick. Low-temperature, blade style conductive tools in conjunction with de-soldering braids can also be used. Once all residual solder is removed, the site should be cleaned with appropriate solvent such as alcohol and a lint-free swab.

4.1.10 Application of Solder Paste

For a precise and uniform solder paste deposition on the redressed site, it is recommended that a miniaturized stencil for the individual component be used. The stencil should be aligned under 50x to 100x magnification, depending on the part. After precise alignment, the stencil should be lowered onto the PCB and the paste should be applied in one pass.

4.1.11 Placement and Attachment of the New Component

The new part should be placed on the site using a split-beam alignment where a dual image of the part leads and the land pattern on the PCB can be viewed on a high-resolution monitor. Once aligned, the part should be placed on the site and attached to the board using the reflow profile used for the part removal.

5.0 Software application information

5.1 Normal mode operation

In normal mode of operation, the baseband processor software runs continuously, providing a navigation solution at the maximum rate of once per second. No power saving functions are applied.

If the power to the module is disrupted, the restart time can be shortened to a warm or hot start by keeping the RTC and SRAM contents valid with a backup battery on the VBATT input.

If the module has been turned off for longer than 2 hours, the unit will revert to a warm or cold start. This is caused by the stored ephemeris data becoming invalid after 4 hours.

5.2 Power management

The TricklePower mode of operation can be enabled to reduce the average power consumption. The main power is supplied to the module continuously. An internal timer wakes the processor from sleep mode. The module computes a navigation position fix, after which the processor reverts to sleep mode. The duty cycle is controlled by a user-configurable parameter.

If ephemeris data becomes outdated, the TricklePower mode will attempt to refresh the data set within every 30 minute period, or for every new satellite that comes into view.

With TricklePower set to a 20% duty cycle, a power saving of 50% can easily be achieved with minimal degradation in navigation performance.

5.2.1 Adaptive TricklePower mode

In Adaptive TricklePower mode, the processor automatically returns to full power when signal levels are below the level at which they can be tracked in TricklePower mode. This is the default behaviour when TricklePower is active. Adaptive TricklePower is always enabled on the Jupiter 32.

5.2.2 Push-to-Fix mode

Unlike TricklePower, the operation in this mode is not cyclic. This mode always forces the GPS software to revert to a continuous sleep mode after a navigation position fix. It will stay in sleep mode until woken by wakeup input, and compute a fresh position.

If the ephemeris data become invalid or new satellites come into view, the RTC has the ability to self activate and refresh the data, thus keeping the restart TTFF very short.

This mode yields the lowest power consumption of the module, and is ideal where a battery powered application requires very few position fixes.

For further information on the TricklePower and Push-To-Fix modes refer to the Low Power Operating Modes application note (LA000513).

5.3 Serial I/O

The module can output serial data in the NMEA format or SiRF Binary format. The serial I/O protocols are defined in the Navman NMEA reference manual (MN000315) and the SiRF Binary Protocol reference manual. These describe the format of the serial data from the module, as well as the structure of the commands. Navman has provided additional functionality to the NMEA protocol as detailed in the following sections.

5.3.1 Default settings

The default protocol is NMEA, at a baud rate of 9600bps. The frame format is 8 data bits, no parity bit and 1 stop bit.

The following messages are output by default, at a rate of once per second each:

$GPGGA, $GPGLL, $GPGSA, $GPGSV, $GPRMC, $GPVTG, $GPZDA

All output messages have checksums by default.

5.3.2 NMEA input commands

All NMEA input commands are in the form:

$PSRFxxx,…[*CS]<cr><lf>

where xxx is a decimal number between 100 and 255 (inclusive).

Note: In each case, CS represents an NMEA checksum. This checksum is detailed in the Navman NMEA reference manual MN000315.

Acknowledgements

All input commands other than the standard SiRF commands will be acknowledged with a sentence of the form:

$PTTK,…*CS

Where the arguments provided to a command are invalid, the response will be the message:

$PTTK,NACK*CS

Checksums

All input commands are accepted whether or not they include a checksum, however if a checksum is included it must be correct in order for the message to be accepted.

Incorrect checksums will result in the response:

$PTTK,CHECKSUM*CS

Unused messages

Input messages where the message ID is not between 100 and 255, or where the message ID does not correspond to a specified function, result in the response:

$PTTK,INVALID*CS

Errors

Errors in message receipt (other than checksum errors) result in the response:

   $PTTK,ERROR,xx*CS

where xx is a hexadecimal error code.

Magnetic Variation (Declination)

The Jupiter 32 module does not calculate the magnetic variation.

5.4 Navman proprietary NMEA low power mode messages

Navman has added a number of proprietary NMEA input messages to configure the TricklePower and Push-To-Fix modes.

5.4.1 Low power configuration

The following message sets the receiver to low power mode:

   $PSRF151,a,bbbb,cccc[*CS]<cr><lf>

where:

Field	Description
a	Push-To-Fix (1=on, 0=off)
b	TricklePower duty cycle (parts per thousand)
c	TricklePower on time (milliseconds)
*Note that Push-To-Fix does not require fields b and c so they may be left blank	

Table 5-1: Low power modes message values

This message is the NMEA equivalent of the SiRF Binary input message ID 151.

System response:

   $PTTK,LPSET,a,bbbb,cccc*CS

The updated values returned by the system are as described in Table 3-1.

5.4.2 Low power acquisition configuration

The following message sets the acquisition parameters of the low power mode:

   $PSRF167,aaaaaa,bbbbbb,cccc,d[*CS]<cr><lf>

where:

Field	Description
a	maximum off time (milliseconds)
b	maximum search time (milliseconds)
c	Push-To-Fix period (seconds)
d	adaptive TricklePower (1=on, 0=off)

Table 5-2: Low power acquisition input values

This message is the NMEA equivalent of the SiRF Binary input message ID 167.

System response:

   $PTTK,LPACQ,aaaaaa,bbbbbb,cccc,d*CS

The updated values returned by the system are as described in Table 3-2.

5.5 Save and Retrieve Configuration Data to Flash

The current settings of the Jupiter 32 will be stored in Flash memory upon a command from the user. Refer to Application Note LA000266. All parameters are written as a set.

The following parameters can be stored and retrieved:

- · Port A and B protocols
- · Port A and B baud rates and frame formats
- · NMEA per-message output rates and checksums
- · SiRF Binary debug and raw tracker data output
- · Datum
- · Altitude hold mode, source and default
- · Degraded mode and timeout
- · DR timeout
- · DOP mode and GDOP/PDOP/HDOP thresholds
- · Elevation navigation mask
- · Power navigation mask
- · Static navigation mode
- · Track smoothing mode
- · TricklePower^{TM} mode, on time and interval
- · Push-to-Fix mode, period, maximum off and acquisition times
- · DGPS source
- · SBAS default PRN

5.5.1 Writing to Flash

The module accepts NMEA messaging or SiRF Binary messaging. The protocols are defined in the *Navman NMEA Reference Manual* (MN000315) and the *SiRF Binary Protocol Reference Manual*. These documents describe the structure of the general commands.

NMEA Store to Flash Memory command

To store all relevant parameters to Flash, send the following NMEA sentence:

$PSRF198[*CS] <cr><lf>

SiRF Binary Store to Flash Memory command

To store all relevant parameters to Flash send a SiRF Binary message with Message ID 198. A payload of this message should have a length of 1 byte.

5.5.2 Retrieving from Flash

At power-up, the unit will retrieve any configuration parameters stored in Flash and restore them as a set. Individual parameters can not be restored. Settings stored in battery backed memory are not over written.

To prevent parameters from being restored, drive the defaults pin GPIO Pad A5 to a logic low.

Logic State	Power-up Action
high or floating	retrieves stored parameters in Flash on power-up if no data in battery backup RAM
low (grounded)	ignores parameters stored in flash

Table 5-3: Actions based on signal state

5.5.3 Reverting to default settings

To restore the default settings, including deleting all parameters stored in Flash, send a factory reset command in either the NMEA or SiRF Binary protocols.

5.6 Selectable User Profiles

The purpose of this feature is to provide some typical application profiles which users can select through a simple NMEA message. These profiles are macro-like commands that modify parameters used for common applications. These profiles produce optimal navigation default settings which conform specifically to an application's requirements. They can be then be saved using the Write-to-Flash feature. The new profile will not be written to Flash automatically.

Available Profiles include:

Default - A general profile that balances the needs of most applications.

Automotive - Takes into consideration higher velocities with occasional stopping.

Pedestrian - Takes into consideration slower velocities.

TricklePower - Power saving mode for battery applications. See Low Power Operating Modes Application Note LA000513

Push to Fix - Only performs a fix when requested. For use in special applications. See Low Power Operating Modes Application Note LA000513

Command

The profiles will be selected by an NMEA command

$PSRF197,n<cr><lf>

where *n* is the profile number:

1. Automotive
2. Pedestrian
3. TricklePower
4. Push to Fix

Refer to Application Note LA000266.

5.7 GPS fix output

The GPS fix output is an active low output on pin 23 of the module. Whenever the unit has either a 2D or a 3D fix, this output will be low. When the unit has no fix, the output will be high.

5.8 Custom application software

The module provides custom modification of the firmware to use available CPU processing capacities and I/O capabilities. The custom application software can be invoked internally or externally, providing the ability to dedicate the module as the host application or an external processor as the host. The module also provides the ability to communicate externally with standard or customer specific communication protocols.

The custom application software has access to the flash memory, navigation data from the GPS Navigation software, and access to various I/O lines used on the module.

This can be developed using the SiRF SDK. The SDK provides all the tools and resources necessary for custom software development. Refer to http://www.sirf.com for more information about the SDK.

6.0 Glossary and acronyms

Axial ratio: For an electromagnetic wave having elliptical polarisation, the ratio of the magnitudes of the major axis and the minor axis of the ellipse described by the electric field vector.

EMI: Electromagnetic Interference

ENIG: Electroless Nickel Immersion Gold

FR4 substrate: Flame Retardant type 4
The usual base material from which plated-through-hole and multi-layer printed circuit boards are constructed. The type '4' indicates woven glass reinforced epoxy resin.

GPS: Global Positioning System
A space-based radio positioning system that provides accurate position, velocity, and time data.

LGA: Land Grid Array
A physical interface for microprocessors. There are no pins on the chip; in place of the pins are pads of bare gold-plated copper that touch pins on the motherboard.

LNA: Low Noise Amplifier
A special type of amplifier used to amplify weak signals captured by an antenna.

MCX: A type of miniature coaxial RF connector

NMEA: National Marine Electronics Association

PCB: Printed Circuit Board

RF: Radio Frequency

SDK: Software Development Kit

SMA: SubMiniature version A connectors are coaxial RF connectors developed in the 1960's as a minimal connector interface for coaxial cable with a screw type coupling mechanism. The connector has a $50\,\Omega$ impedance.

SRAM: Static Random Access Memory

SMT: Surface mount technology (SMT) is a method for constructing electronic circuits in which the components are mounted directly onto the surface of printed circuit boards

TTFF: Time-To-First-Fix
The actual time required by a GPS receiver to achieve a position solution. This specification will vary with the operating state of the receiver, the length of time since the last position fix, the location of the last position fix, and the specific receiver design.

Push-To-Fix and TricklePower are registered trademarks of SiRF Technologies.

Reader response:
Navman strives to produce quality documentation and welcomes your feedback. Please send comments and suggestions to tech.pubs@navman.com. For technical questions, contact your local Navman sales office or field applications engineer.