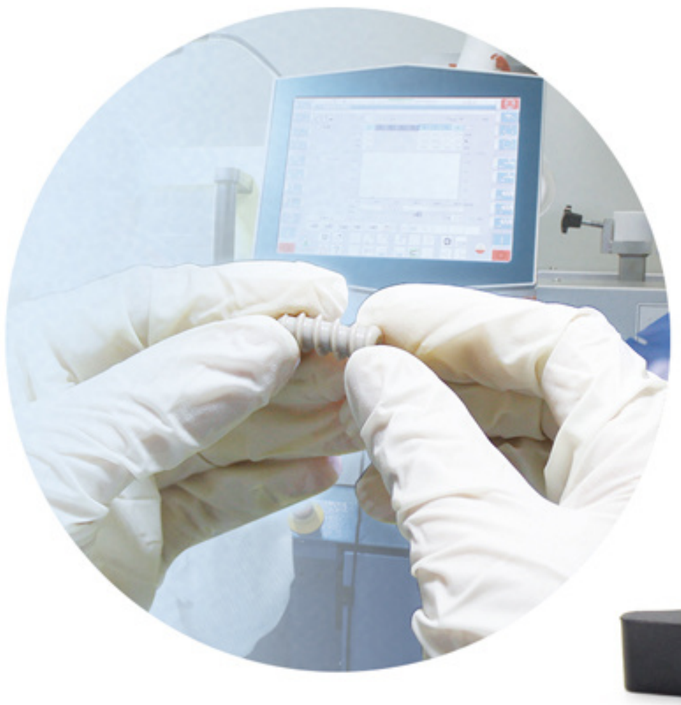
君华股份
JUNHUA PEEK

Mastering PEEK Injection Molding: From Process Control to Defect Resolution



江苏君华特种高分子材料股份有限公司
Jiangsu Junhua HPP Co., Ltd.

www.junhuaPEEK.com

关于我们

www.junhuaPEEK.com

ABOUT US

50+

Patents for inventions

26 invention patents and 34 utility model patents have been granted, including《Preparation method of phenol-terminated polyaryletherketone polymers》and《A continuous CF/PEEK thermoplastic composite material and its preparation method》

Company Profile

JiangSu JunHua HPP Co.,Ltd. specializes in the application, development and production of PEEK (poly-ether ether ketone) , PPS (Poly-LRB-p-phenylene sulfide)PI, PI (polyimide) and other high performance special engineering plastics resins and profiles, and the formation of a PEEK resin with raw materials polymerization, modified granulation, such as continuous extrusion of bar and tube sheet profiles and finished parts of the in-jection molding and machining of the whole industry chain.

Junhua has a provincial industrial design center, team of more than 10 professional designers and nearly 30 professional mold manufacturing team members. Our team's comprehensive ability covers product design, mold design, manufacturing and testing and other aspects, each set of mold development are subject to rigor-ous review to provide protection for the enterprise's professional production. We have 18 years of PEEK injec-tion molding experience, as well as the mature process and supporting equipment, to ensure that the appear-ance and size of the product to meet customer needs, while minimizing the loss of raw materials. In addition, we provide mold hosting and maintenance services, including regular inspection, cleaning and maintenance to extend the life of the mold, and to ensure the stability and consistency of the injection molding process.

Welcome to our guidance exchange!

Core Advantages

Over 18 Years
High temperature special engineering plastic mold manufacturing experience

Twenty-two
Imported and domestic injection equipment with different injection tonnage

Nearly 30
Professional design and mold manufacturing team members

Over 3,000 Sets
Experience in mold development

Level 10,000
Injection Molding Workshop

15000m²
Standardized plant (60000m² under construction)

君华股份
JUNHUA PEEK

目录 / CONTENTS

Mastering PEEK Injection Molding: From Process Control to Defect Resolution

(1) Screw Selection

Most screws are generally suitable for processing PEEK material. Prior consideration should be given to:

1.Long diameter ratio for the screw, typically between 18 1 to 24 1.

2.Compression ratio of 2:1 or 3:1.

3.Check valve should be installed at the head of the screw.

4.Consider using a double alloy material scree to prevent the molded product from being brittle.

(2) Injection Volume

Due to the high forming temperature of PEEK particle, the dwell time should be as short as possible. Each injection volume should be more than 30% of the machine' s maximum shot volume. If using a machine with a larger injection volume, it is necessary to appropriately reduce the temperature of each section of the barrel.

(3) Molding Process

Injection pressure: 50-100Mpa for pure material; 70-140Mpa for material with fibers

Injection speed: Medium to high speed.

Holding temperature: 40-100Mpa.

Back pressure: 0.2-5Mpa

Screw speed: 50-100r/min

Cooling time: Based on specific product requirements.

(4) Drying

PEEK particles should be thoroughly dried before injection molding and enhance product performance. The moisture content after drying should be less than 0.04%w/w. Drying should start when the equipment reaches the set temperature value, and it is recommended to use:

Drying Equipment	Drying Process	(Note: The thickness of the raw material in the pallet should preferably be no more than 25mm, with sufficient air flow channels.)
Electric heat shoe type drying oven	160℃ for 6 hours, 180℃ for 4 hours	
Hot air circulation oven	140℃ for 4 hours, then 160℃ for 2 hours	
Dehumidifying dryer (for moisture content of -40°)	140℃ for 4 hours, then 160℃ for 3 hours	

(5) PEEK Reference Barrel Temperature

Grade	Back ℃ Temperature	Middle ℃ Temperature	Front ℃ Temperature	Nozzle ℃ Temperature	
PEEK5600G	350-365	360-370	370-380	375	(Note: the dwell time of particle should be less than 1 hour in barrel with temperature 360 , if need more should reduce the barrel temperature down to 340℃ .)
PEEK5600GF30	360-380	360-380	370-390	375	
PEEK5600CF30	360-385	370-390	380-395	395	
PEEK5600LF30	350-375	360-380	370-390	380	

(6) Mold Temperature

Mold Heating Methods	Electric heating plate, oil temperature control
Mold Temperature	180℃-220℃
(Note: Mold values refer to the surface temperature of the mold, not the setting temperature of the control unit. A high mold temperature allows the product to crystallize sufficiently during the molding process to enhance the properties and appearance of the product. If high mold temperature cannot be used during the molding process, the products should be post-treated, the post-treatment condition is 200 /hour, and the post-treatment will bring secondary deformation and other problems.	

(7) Product Defect Causes and Their Solutions

Product Defect	Causes	Solutions
Dark and Transparent Edges 	Low mold temperature	Increase mold temperature
Brittle Product 	1. barrel temperature is too high 2. product internal stress is too large 3. raw material stagnation barrel time is too long 4. raw material is not clean or mixed with other raw materials 5. fusion line is not well combined 6. excessive use of release agent 7. product crystallinity is not enough	1. reduce the barrel temperature 2. increase the release section of the pressure preservation or the use of the oven for heat treatment to release the stress 3. shorten the cycle of molding 4. check the raw materials 5. increase the temperature of the mold, increase the injection and pressure preservation Increase the mould temperature, injection and holding pressure, and injection speed 6.Reduce the usage amount 7.Increase the mould temperature or use the oven for heat treatment.

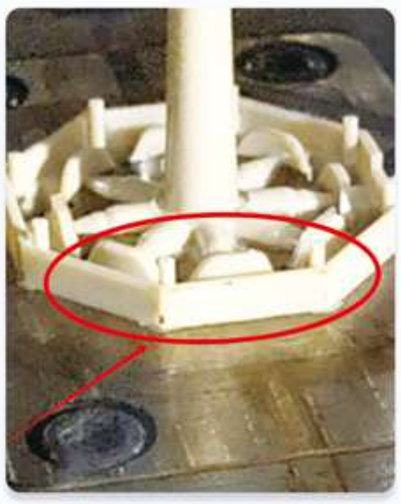
Product Defect	Causes	Solutions
Incomplete Injection 	1.Low barrel temperature 2.low injection temperature 3.insufficient amount of sol 4. injection time is too short 5. injection speed is too slow 6. mold temperature is too low 7. mold temperature is not uniform 8. mold exhaust bad 9. nozzle blockage 10. injection of glue is not uniform 11. runner or gate is too small 12. screw thrust ring is worn out 13. machine glue amount is not enough	1. Increase the barrel temperature 2. Increase the injection pressure 3. Increase the amount of solvent 4. Increase the injection time 5. Increase the injection speed 6. Increase the temperature of the mold 7. Check the heating rods or the oil circuit for damage or blockage 8. Increase the proper air venting 9. Disassemble the nozzle and clean it up or replace it with a new one 10. Optimize the structure of the product and increase the overflow groove 11. Replace the size of the mold runner or the size of the sprue 12. Disassemble it, inspect it, repair it and replace it 13. Replacement of larger models
Cold Material 	1. too high barrel temperature 2. too much internal stress in the product 3. too much time for the raw material to stay in the cylinder 4. unclean raw material or mixed with other raw materials 5. poor bonding of the fusion line 6. excessive use of release agents 7. insufficient crystallinity of the product	1. Reduce barrel temperature 2. Increase holding pressure release section or use oven for heat treatment to release stress 3. Shorten the molding cycle 4. Check the raw material 5. Increase the mold temperature, injection and holding pressure, and injection speed 6. Reduce the amount of use 7. Increase mold temperature or use oven heat treatment
Sink Mark 	1. low injection pressure and speed 2. short holding time 3. low holding pressure 4. irrational design of gate or runner 5. short injection time 6. nozzle clogging 7. material temperature is too high 8. improper temperature of the mold 9. bad exhaust of the mold 10. wall thickness of the product is not the same thickness 11. screw non-return valve is worn out or damaged 12. the machine is too big (the material pipe is too big) 13. insufficient amount of sol	1. Increase the injection pressure and speed 2. Increase the pressure holding time 3. Increase the pressure holding capacity 4. Adjust the mould gate, runner size or position 5. Increase the injection time 6. Disassemble and clean up or replace 7. Decrease the material temperature 8. Adjust to the appropriate temperature 9. Set up venting holes in the shrinkage 10. Increase the injection pressure or optimize the structure of the product 11. Demolition and replacement 12. Replacement of the machine (replacement of the smaller specification of the material pipe) 13. Increase sol-gel metering stroke

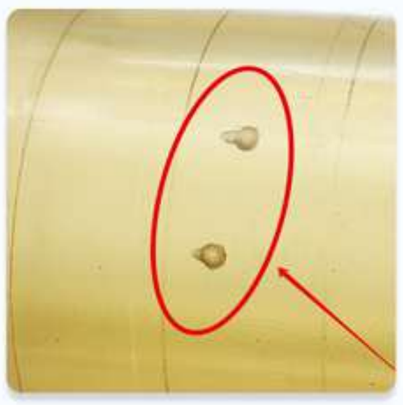
Product Defect	Causes	Solutions
Crack 	1. Overheating of material 2. Moisture in material 3. Dead spots in sol barrel 4. Excessive use of release agent	1. Reduce barrel temperature, reduce nozzle humidity, shorten molding cycle, reduce injection speed, reduce screw speed 2. Dry material 3. Clean screw, nozzle and barrel
Air Trap and Galling 	1. Mold exhaust design is unreasonable 2. injection speed is too fast 3. injection pressure is too large 4. gate design is unreasonable	1. Reduce the injection pressure 2. Reduce the injection speed 3. Improve the cavity venting 4. Change the location, size and type of injection ports
Flashing 	1. barrel or mold temperature is too high 2. raw material pollution 3. high liquidity of raw materials 4. clamping force is too low 5. injection pressure is too high 6. holding pressure is too high 7. injection speed is too fast 8. barrel temperature is too high 9. injection molding machine template deformation 10. mold venting holes are blocked 11. mold wear and tear 12. sprue, runner size or position is not appropriate 13. mold core eccentricity 14. mold deformation 15. improper selection of the type of injection molding machine	1. lower the barrel temperature or mold temperature 2. check the raw material to find out the source of contamination 3. replace the raw material or lower the barrel and mold temperature 4. increase the clamping pressure 5. lower the injection pressure 6. Reduce holding pressure 7. lower the injection speed 8. lower the barrel temperature or adjust the screw speed 9. overhauling 10. checking and cleaning 11. repairing or replacing the wear parts 12. optimize the gate, runner size and location 13. re-molding increased Reinforcement positioning 14.Overhaul or replacement 15.Selection of matching injection molding machine
Product Twisted or Deformed 	1. The product has not been cooled when ejected 2. Uneven shape and thickness of the finished product 3. Uneven feeding of the cold material wells 4. Unevenness of the ejector system 5. Excessive filler 6. Uneven mold temperature	1.Reduce the temperature of mold and cylinder, prolong the cooling time 2.Do the shaping tooling, modify the product structure 3.Modify the size and position of the cold material well 4.Improve the ejector system 5.Reduce the injection pressure, speed, time and the amount of material used 6.Check the heating rods whether they are damaged or under-installed and leakage, and whether the oil circuit is clogged.

Product Defect	Causes	Solutions
Jetting, Flow Marks 	1. Poor melting of raw materials 2. Mold temperature is too low 3. Injection speed is too slow or too fast 4. Injection pressure is too high or too low 5. Raw materials are not clean or mixed with impurities 6. Cold material well is too small 7. Thickness of the finished product bonding surface is too big a difference 8. Holding time is too short 9. Gate is too small or improperly located 10. Raw materials are not clean or mixed with other materials	1. Increase raw material temperature and back pressure or speed up screw speed 2. Increase mold temperature 3. Adjust proper injection speed 4. Adjust proper injection pressure 5. Check raw material 6. Increase the cold material well 7. Change the design of finished product or the location of the cold material well 8. Increase the holding time 9. Increase the gate or change the location 10. Check raw material
Excessive Shrinkage 	1. unreasonable setting of molding process 2. too small gate 3. big difference between shrinkage of raw material and actual 4. unreasonable design of molds	1.Reduce the mold temperature 2.Increase the injection pressure 3.Increase the holding pressure 4.Increase the gate size 5.Change the raw material 6.Optimize the design structure
Silver Streaks 	1. the raw material contains water 2. the barrel temperature is too high or the mold temperature is too high 3. too many additives in the raw material or uneven mixing 4. uneven thickness of the material 5. air trapped in the material tube 6. improper flow rate of the raw material in the mold cavity 7. the mold temperature is too low	1. Dry the raw materials, increase the back pressure 2. Reduce the cylinder temperature or mold temperature 3. Reduce the additives or mix them evenly 4. Choose the raw materials with uniform particles 5. Increase the back pressure 6. Adjust the size and position of the cold material well 7. Increase the mold temperature
Weld Line 	1. Poor melting of raw materials 2. Mold temperature is too low 3. Injection speed is too slow 4. Injection pressure is too low 5. Raw materials are mixed 6. Too much release agent is used 7. Gates are too small or improperly positioned 8. Poor mold venting 9. Nozzle of the mouth of the material is too small 10. Material curing is too fast 11. Raw materials are unclean or mixed with other materials 12. Raw materials have poor liquidity	1. Increase the barrel temperature or increase the back pressure 2. Increase the mold temperature 3. Increase the injection speed 4. Increase the injection pressure 5. Check the raw materials 6. Use less or no release agent 7. Modify the mold or adjust the location of the gate 8. Increase the exhaust groove 9. Modify the gate of the mold or replace the nozzle 10. Increase the cold material well 11. Check the raw materials 12. Increase the temperature of the barrel, mold or increase the auxiliary materials

Product Defect	Causes	Solutions
Dark Spots 	1. the raw material heating stay time is too long carbonation or temperature is too high 2. the raw material is mixed with other foreign matter 3. there are dead corners in the cylinder 4. scorched spots when shooting into the mould 5. drying equipment is not cleaned up	1. Shorten the residence time of raw materials, reduce the heating temperature 2. Check the raw materials 3. Reduce the injection pressure and speed 4. Check whether there is any gap or corrosion on the contact surface between the injection nozzle and the flange 5. Check the drying equipment
Cracking or deepening of the finished product during mould opening or ejection 	1.Overfilling 2.Mould temperature is too low 3.Part of the release angle is not enough 4.There is a release chamfer 5.Ejector imbalance 6.Ejector stroke is not enough or mould is not appropriate 7.Vacuum phenomenon of the product when removing the mould	1.Decrease the injection pressure, time, speed and the amount of injected glue 2.Increase the temperature 3.Overhaul and check the mould 4.Overhaul and check the mould 5.Overhaul and check the mould 6.Extend the ejector pin or overhaul the mould 7.Slow down the ejector speed or increase the air intake structure on the mould.
Scratch Mark 	1.Mould cavity has inverted buckle 2.Mould demoulding slope is too small 3.Mould cavity surface is rough 4.Product structure design is unreasonable 5.Ejection is not smooth	1.Overhaul the mould 2.Increase the demoulding slope 3.Mould cavity polishing 4.Optimize the product structure 5.Adjust the way of ejection
Porosity, Insubstantial Density 	1.Raw material moisture content exceeds the standard 2.Barrel temperature is too high so that the raw material decomposition 3.Injection speed is too fast 4.Cooling time is too long 5.Back pressure is not enough 6.Insufficient injection 7.Insufficient amount of molten rubber 8.Running channel, gate is too small 9.Pressure holding time is too short, pressure is not enough 10.Injection time is too short 11.Product wall thickness, bar or column is too thick	1. Dry the raw material thoroughly 2. Reduce the barrel temperature 3. Slow down the injection speed 4. Reduce the cooling time of the mould 5. Increase the back pressure 6. Check the barrel temperature injection press injection time 7. Increase the amount of solvent 8. Increase the size of the runner and gate 9. Increase the holding time, increase the holding pressure 10. Increase the injection time 11. Optimize the product structural design or the size and position of the gate

Product Defect	Causes	Solutions
Shrinkage Hole 	1.Product wall thickness is too thick 2.Product structure design is unreasonable 3.Injection pressure is small 4.Holding time is too short 5.Injection speed is too slow 6.Injection port size is small 7.Holding pressure is too small	1.Product structure to do the design of glue reduction 2.Optimize the product structure to make the product wall thickness is proportional 3.Increase the injection pressure 4.Extend the holding time 5.Increase the injection speed 6.Modification of the size of the injection port 7.Increase the holding pressure
Gates and Sprue Sticking to the Mould 	1. Injection pressure is too high 2. gate runner is too big 3. gate cooling temperature is not enough 4. runner demoulding angle is not enough 5. inside surface of the runner is not light or there is demoulding chamfer 6. inside of the sprue set is rough or there is no pulling hook 7. R angle of the sprue set does not match with the injection nozzle	1. reduce injection pressure 2. modify the mould 3. prolong the cooling time to reduce the temperature of the cylinder 4. modify the demoulding angle 5. modify the demoulding angle, overhaul the mould polishing 6.Replace the new sprue or increase the size of pulling hook or ball head 7.Re-adjustment and matching
Product Surface is not Smooth 	1. Mould temperature is too low 2. Insufficient dosing of raw materials 3. Too much release agent in the mould 4. Mould cavity surface is not smooth	1. Increase the mould temperature 2. Increase the injection pressure, time and amount of material used 3. Wipe and clean and check if it's clean 4. Polishing the surface of moulds
Black Stripe/Mottled 	1. barrel temperature is too high 2. screw speed is too fast 3. screw deformation or damage and barrel local friction heat 4. nozzle temperature is too high or aperture is too small 5. barrel or machine is too large 6. nozzle inside the residue 7. nozzle hole deformation caused by shear heat is too high	1. reduce the heating temperature 2. reduce the speed of the screw 3. dismantle the overhaul or replace it 4. re-adjustment of the temperature and aperture diameter 5. replacement of the barrel or machine 6. thoroughly Clean the nozzle or replace it with a new one 7.Take it out for overhaul or replacement

Product Defect	Causes	Solutions
Product Sticky Mould 	1. Packing pressure or barrel temperature is too high 2. Holding time is too long 3. Injection speed is too fast 4. Uneven feeding makes part of the oversaturation 5. Cooling time is not enough 6. Mould temperature is too high or too low 7. There is a release chamfer in the mould 8. Multi-cavity mould inlet is not in a balanced position or the size of the inlet of single-cavity mould is not reasonable 9. Unreasonable design of the demoulding and exhausting of the deep cylinder parts 10. Holding pressure is too high 11. Misaligned mould core 12. Surface is too rough	1.Reduce the injection pressure or barrel temperature 2.Reduce the holding time 3.Reduce the injection speed 4.Change the size or location of the cold material well 5.Increase the cooling time 6.Adjust the mould temperature 7.Repair the mould to remove the chamfer 8.Optimize the gate size and location as close as possible to the main stream 9.Provide sufficient air escape 10.Decrease the holding pressure 11.Adjust the core to increase the rib positioning 12.Polishing of the mould cavity
Chip 	1. Raw materials are dirty or mixed with other materials 2. Recycled too often 3. Contains too much PTFE	1. Check the raw materials 2. Recycled at least 70% of new materials 3. Reduce the content of PTFE
Discoloration 	1. raw materials are not clean or mixed with other materials 2. mould temperature is low 3. barrel temperature is too high 4. raw materials stay in the barrel for a long time 5. injection moulding machine barrel is too large 6. different batches of different grades of raw materials mixed 7. switching colours did not clean up the hopper, the barrel	1. Check the raw material 2. Increase the mould temperature 3. Reduce the barrel temperature 4. Shorten the moulding cycle 5. Replace the small barrel injection moulding machine 6. Check the raw material 7. Check the clean up

Product Defect	Causes	Solutions
Bubbles 	1. raw material is not dry enough 2. injection speed is too fast 3. injection pressure is too big 4. mould exhaust bad 5. sol barrel mixed with air 6. raw material is not pure mixed with other raw materials or debris	1. dry the raw material adequately 2. reduce the injection speed 3. reduce the injection pressure 4. clean up the mould and increase the exhaust holes 5. increase the back pressure, reduce the temperature of the barrel 6. check the raw material
Carbonation 	1.impure raw materials mixed with other raw materials or debris 2. barrel is not completely cleaned up 3. there are dead ends in the barrel 4. barrel temperature is too high so that the decomposition of raw materials 5. raw materials in the barrel to stay too long 6. barrel is too large	1. Check the raw material 2. Thoroughly clean the barrel 3. Check whether there is a gap between the nozzle and the flange 4. Reduce the temperature of the barrel 5. Shorten the moulding cycle 6. Replace the small melting cylinder
Fiber Emergency on the Surface 	1. Low barrel temperature 2. Low mould temperature 3. Slow injection speed 4. High melt volume 5. Uneven wall thickness 6. Unreasonable structure design of gate size	1. Increase the barrel temperature 2. Increase the mould temperature 3. Increase the injection speed 4. Decrease the amount of melt 5. Try to make the wall thickness of the molded part uniform in all places and avoid sharp corners and gaps to ensure smooth flow of melt. 6. Optimize the gate structure and size
Delamination 	1. metal parts not do preheating treatment 2. metal parts not do overmoulding structure 3. mould or metal parts temperature is low 4. injection or holding pressure is low 5. injection or holding pressure time is short 6. PEEK layer and metal parts cooling shrinkage	1. injection overmoulding before the metal parts need to do preheating treatment 2. briefly see see to do the structure of the anti-detachment 3. to increase the temperature of heat treatment of the mould or metal parts 4. improve the pressure of the injection or holding 5. Increase the injection or holding pressure time 6.The second heat treatment is needed immediately after injection moulding and overmoulding.

(8) PEEK Common Injection Molding Products

PEEK Applications in the Textile Printing and Dyeing Machinery Industry

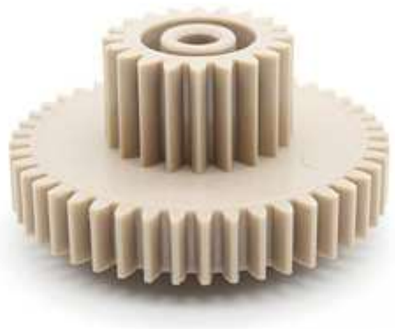
- PEEK Amplitude Adjusting Nut / PEEK Hexagonal Shaft Sleeves

PEEK Application in the Medical Industry

- Sports Medicine: PEEK Anchors

PEEK Application in the Gear Drive Control Seal Industry

- PEEK Gears



- PEEK Thrust Washers

PEEK Application in the Electronic Semiconductor Industry

▼ PEEK Coil Skeletons / PEEK Wafer Clamps

PEEK Application in the Analytical Biochemical Industry

▼ PEEK Connectors / Union

PEEK Application in the Aerospace Military Industry

▼ GF/PEEK High Transparency Radome

▼ PEEK Tube Frames/Clamps

Official Wechat account

Customer service account

诚信·务实·合作·创新·共赢

INTEGRITY · PRAGMATISM · COOPERATION · INNOVATION · WIN-WIN SITUATION

JiangSu JunHua HPP Co.,Ltd.

Tel: 0519- 86228816　13915088386

Fax: 0519- 86228826　86228876

Web: www.junhuaPEEK.com

Mail: chinaPEEK@chinaPEEK.com

International Business Department

Tel: +86-519-8622 8823

+86-519-8622 8851

Mail: chinaPEEK@chinaPEEK.com

Add: South Wuyi Road 377#, Creation industrial park, No.11 workshop, Wujin Hi-tech District, Changzhou, China