Design of injection molding machine tie rod stress groove optimized based on SolidWorks Simulation

Zhang Yuanbo, Zhang Wenjun, Yan Houming

(Donghua Machinery Co. LTD., Dongguan 523118, Guangdong , China)

Abstract: The tie rod is the main load-bearing component of the clamping part of an injection building machine, typically possessing high strength and rigidity to withstand various forces such as mold expansion during the injection molding process. However, in the production of plastic products, especially during the high-speed production of thin-walled fast food containers, cutlery, and other items, fracture is prone to occur, resulting in significant economic losses for both customers and injection molding machine manufacturers. This paper establishes a finite element analysis model for the entire tie rod and nut assembly, analyzes and optimizes the stress concentration issues near the stress groove of the tie rod and the threads of the nut, obtains the dimensional parameters of the tie rod, and accumulates experience in the optimal design of the tie rod stress groove.

Key words: tie rod; injection molding machine; stress groove; finite element analysis; thread

Classification number: TQ320.66 **Article number:** 1009-797X(2026)10-0040-07

Document code: B **DOI:** 10.13520/j.cnki.rpte.2026.10.015

0 Background

In the clamping system of an injection molding machine, the tie bar primarily serves to connect the moving platen and the stationary platen, as well as provide support and guidance. The tie bar typically possesses high strength and rigidity to withstand various forces during the injection molding process, such as the clamping force. Its structural design requires consideration of multiple factors, including strength, stability, and precision.

During the COVID-19 pandemic, due to regional lockdowns and joint prevention measures, schools, factories, grid personnel, volunteers, and others mainly engaged in meal distribution, leading to a surge in the use of disposable plastic fast food containers. The demand for injection molding machines and molds for producing plastic meal containers also exceeded supply. However, due to the relatively thin wall thickness (0.4 mm), large flow length (300~400), multiple mold cavities (4~8 cavities), and fast production cycle (5~6 s) of disposable plastic fast food containers, in this production environment with high clamping force and efficiency pursuit, fatigue fracture is prone to occur at the first and second tooth roots connected by the tie rod and nut, as shown in Figure 1. Especially during the pandemic, due to regional lockdowns and joint prevention measures, maintenance engineers face inconveniences in traveling and parts transportation, and customer production halts can result in significant losses.



Figure 1 Physical image of the failed tie rod of an injection molding machine

To this end, injection molding machine manufacturers producing high-speed thin-walled products have invested a significant number of design and research personnel to

Biography: Zhang Yuanbo (1983-), holds a bachelor's degree and works as an engineer, primarily engaged in the design and application of injection molding machines.

tackle the problem. Many measures have been taken from the design perspective, such as modifying the material and heat treatment of the tie rod, adopting a suspended structure for the nut, enlarging the fillet of the tie rod transition section, using hobbing for the thread, and quantitatively designing the diameter of the stress groove. These measures have indeed achieved certain effects in practical use, improving the reliability of the tie rod's operation. However, fracture phenomena still occur frequently. Some injection molding machine manufacturers simply thicken the tie rod of the injection molding machine, which does solve the problem of tie rod fracture to a certain extent. However, due to frequent fractures in old machines at customers' locations, maintenance costs remain high, and the machines can only be used to reduce the clamping tonnage, which also has a negative impact on the injection molding machine manufacturers.

This article focuses on optimizing the design of the stress groove structure of the tie rod through finite element analysis. By doing so, it effectively reduces the probability of tie rod failure in injection molding machines without altering the tie rod diameter, which is of great significance for the quality and reliability of the tie rod. Additionally, optimizing the design of the tie rod stress groove has a significant impact on the stability of the clamping structure of the injection molding machine.

1 Injection molding machine tie rod structure and stress analysis

1.1 Calculation of structural strength of tension rod

When designing the tie bars of an injection molding machine, it is generally assumed that the four tie bars equally bear the clamping force. The tensile stress generated by the tie bars is approximately a static load, and their tensile strength, fatigue strength, and thread strength are verified based on the tie bar diameter.

Since this article focuses on the optimal design of stress grooves through finite element analysis under the condition of constant rod diameter, the calculation will not be expanded. Instead, the emphasis will be placed on the comparative analysis before and after the optimization of the stress grooves.

1.2 Finite element analysis method

1.2.1 3D modeling

Since the focus of this article is on stress analysis and optimization of the tension rod, referring to common fracture locations, 3D models of the tension rod and nut were quickly established using SolidWorks.

To reduce the number of finite element meshes, the circular hole on the nut end face used for rotation is removed, while the necessary circular corners for load-bearing are retained. The contact surface between the triangular external thread of the tension rod and the internal thread of the nut is a spatial helical surface, and the force acting between them is a spatial general force system. Due to the small lead angle of the thread, the influence of the thread lead angle can be neglected in the calculation model. A series of circumferential annular flanges with standard thread forms are used to replace the continuous thread. Since the load and constraint conditions of the threaded connection part of the tension rod are also axisymmetric, the threaded connection of the tension rod is simplified to an axisymmetric problem, which is suitable for the integration of finite element analysis and thread parameter optimization.

Due to the simplification into an axisymmetric problem, in order to reduce the number of finite element meshes, after the tie rod and nut are assembled as a whole, the cylinder is divided into 36 equal parts, and one of them is selected for finite element analysis. At the same time, the tie rod is cut off at a distance of 100 mm from the stress groove, retaining the end connected to the nut, as shown in Figure 2.

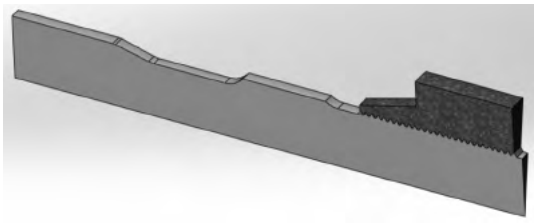


Figure 2 Simplified 3D model diagram

1.2.2 Example setting

Select static analysis, and choose "FFEPlus" as the solver.

The tension rod is made of 40Cr steel with quenching and tempering treatment, and the nut is made of 45 steel with quenching and tempering treatment. The elastic modulus is 206 GPa, and the Poisson's ratio is 0.3. For the tension rod: tensile strength is 685 MPa, yield strength is 490 MPa; for the nut: tensile strength is 665 MPa, yield strength is 345 MPa.

Select "No Penetration" for the contact type option of the component contacts, apply symmetric constraints to the four cut surfaces, and also apply symmetric constraints to the left end surface of the pull rod.

The clamping force of the machine is 3600 kN, with each of the four tie rods bearing a normal clamping force of 900 kN. During modeling, a 1/36 section was taken, meaning the force-bearing end surface of the nut bears a normal force of 25 kN.

The grid type is selected as the solid standard grid. For more accurate calculations, grid control is applied to all 208 surfaces, including threaded tooth surfaces and fillets, with a unit size defined as 2 mm and a ratio defined as 1.5; the unit size for the remaining parts is defined as 6 mm, with a tolerance of 0.3 mm; the 3D grid division result is shown in Figure 3. Total number of nodes: 76,440; total number of units: 46,986; maximum aspect ratio: 13.144; the proportion of units with an aspect ratio < 3 is 94.4%; the proportion of units with an aspect ratio > 10 is 0.0106%; the proportion of distorted units is 0. It is a high-quality grid.

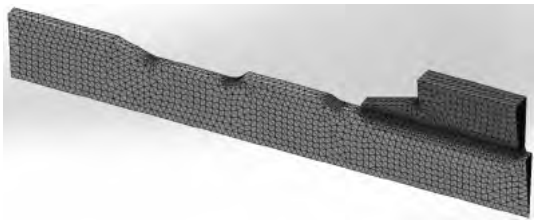


Figure 3 Grid division results

Then run the finite element analysis and wait for a few minutes to obtain the results.

2.3 Calculation results

The stress distribution results of the initially designed tie rod are illustrated in Figures 4 to 7:



Figure 4 Cloud chart of comprehensive stress distribution of the tension rod

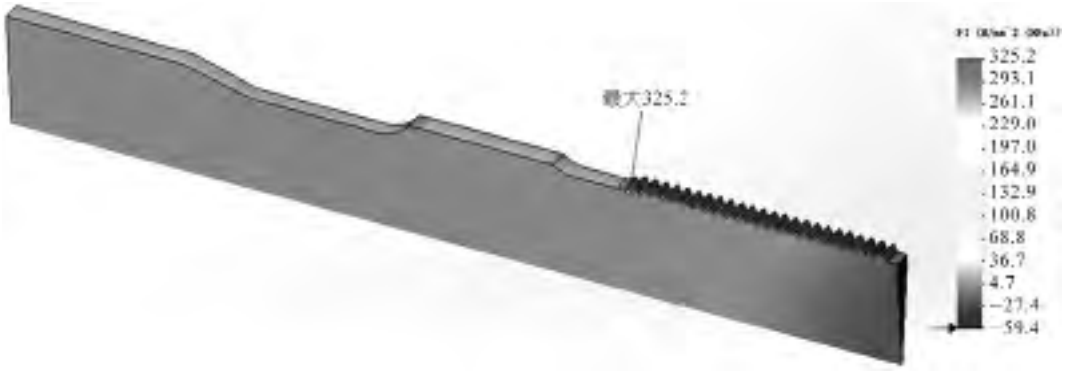


Figure 5 Cloud chart of the distribution of principal stress P_1 on the tension rod

The maximum combined stress of the pull rod is 273.7 MPa, occurring at the root of the load-bearing tooth of the first complete external thread; the principal stress P1 is 325.2 MPa, located at the same position as the maximum combined stress. Both values are less than the material yield strength of 490 MPa.

The maximum comprehensive stress of the nut is 181.3 MPa, occurring at the root of the load-bearing tooth of the first complete internal thread; the principal stress P1 is 209.9 MPa, located at the intersection fillet between the left end face and

the bevel. Both values are less than the material yield strength of 345 MPa.

Based on the analysis results, the location where the maximum comprehensive stress occurs coincides with the actual fracture failure position of the tension rod, indicating that the trend of stress analysis is effective. Therefore, the primary direction for the optimization design of the tension rod should be to reduce comprehensive stress and shift the location of stress concentration.

2 Optimized design of stress groove based on SolidWorks simulation

2.1 Optimization design approach and modeling

The stress groove of the tension rod primarily serves to transfer the load-bearing capacity at the thread root to the stress groove. Generally, the smaller the diameter of the stress groove (d_1), the greater the unloading capacity of the tension rod. Additionally, the width L, angles θ_1 and θ_2, as well as the radii R_1 and $_{R2}$ of the inclined surface and the bottom of the stress groove, all have an impact on the unloading capacity of the tension rod. Among these factors, the angle θ_2 is commonly set to 15° or 20° according to enterprise design specifications, considering the installation of a dust seal ring for the tension rod; while θ_1 is set to 45°. The dimensions are shown in Figure 6.

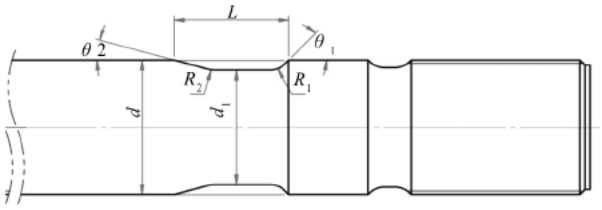


Figure 6 Dimensions of the stress groove of the tension rod

To identify the optimal dimensions of the tension rod stress groove, an analysis model was established as shown in Table 1.

2.2 Select optimization results based on finite element analysis

Based on the dimensions listed in Table 1, a combination design was conducted. The extracted results of the finite element analysis are presented in Table 2 and Figures 7-9.

Table 1 Finite element analysis model for the dimensions of the tension rod stress groove

Pull rod diameter	Stress groove diameter	Diameter ratio	Stress groove width	Fillet 1	Fillet 2	Bevel angle 2
d/ mm	d_1/ mm	d_1/d	L/ mm	R_1/ mm	R_2/ mm	θ_2/°
130	120	0.92	100	15	30	15
	110	0.85	110	20	40	
	100	0.77	120	25	50	
	90	0.69	130	30	60	20
	80	0.62	140	35	70	
	70	0.54	150	40	80	

Table 2 Finite element analysis model and results for the stress groove dimensions of the pull rod

Pull rod diameter	Stress groove diameter	Diameter ratio	Stress groove width	Fillet 1	Fillet 2	Bevel angle 2	Maximum comprehensive stress
d/ mm	d_1/ mm	d_1/d	L/ mm	R_1/ mm	R_2/ mm	θ_2/°	MPa
130	120	0.92	100	15	30	15	271.2
	110	0.85	110	20	40		282.3
	100	0.77	120	25	50		274.2
	90	0.69	130	30	60	20	270.5
	80	0.62	140	35	70		271.8
	70	0.54	150	40	80		280.2

Through comparative analysis of the results, it was found that although the maximum comprehensive stress changed its location when the diameter of the stress groove d1 was reduced to 70 mm, the magnitude of the comprehensive stress did not change significantly, indicating that this optimization design was unsuccessful. Therefore, we changed our approach and incorporated the size of the thread relief groove on the right side into the optimization consideration. The design of dual stress groove size optimization is shown in Figure 10. Where θ_3 and θ_4 are 45°, and R_3=R_4.

Figure 10 Optimized dimensions of double stress grooves on the tension rod

After multiple combinations optimization and process dimension optimization, the final finite element analysis results



Figure 7 Cloud chart of stress results from finite element analysis of a stress groove with a diameter of 110 mm



Figure 8 Stress contour plot of the finite element analysis for a stress groove with a diameter of 90 mm



Figure 9 Cloud chart of stress results from finite element analysis of a stress groove with a diameter of 70 mm

Table 3 Finite element analysis model and results of the stress groove dimensions of the pull rod before and after optimization

	Pull rod diameter d/mm	Stress groove diameterd1 (d_3)/mm	Diameter ratio d_1/d	Stress groove width 1 L/ mm	Stress groove width 2 L_1/ mm	All rounded corners R/ mm	Bevel angle 2 θ_2/°	Maximum comprehensive stress of the tension rod/MPa	Maximum principal stress P1 of the tension rod/MPa
Before optimization	130	111(122)	0.85	100	43	20	15	273.7	326.2
optimized	130	112(112)	0.86	100	40	20	15	170.2	197.8

are as shown in Table 3.

From the analysis results, it can be seen that after optimization, the comprehensive stress of the pull rod decreased from 273.3 MPa to 170.2 MPa, a decrease of about

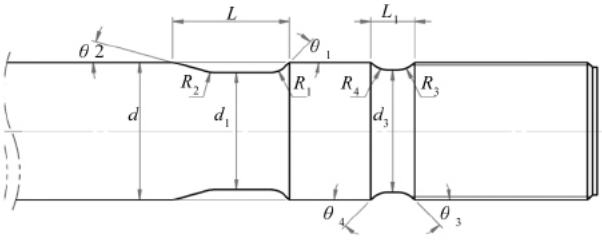


Figure 10 Optimized dimensions of double stress grooves on the tension rod

38%; the principal stress P1 decreased from 326.2 MPa to 197.8 MPa, a decrease of about 40%; the optimization effect is quite evident. The results indicate that applying multiple sets of dimensional parameters for integration in finite element analysis makes the design optimization of stress grooves more reasonable.

By extracting the cloud chart of the comprehensive stress of the tension rod exceeding 100 MPa, it can be visually observed that, apart from the bottom fillets of the first eight threaded sections that bear the load, there are distinct stress bands at the bottom and fillets of the first stress groove near the thread, as well as at the fillets of the second stress groove. By utilizing the probing tool of SolidWorks Simulation to detect the stress distribution in the first stress groove, it can be visually observed that the comprehensive stress at the junction of the groove fillet R3 and groove d3 is 133.3 MPa, which is lower than the material yield strength of 490 MPa.

Some results are shown in Figures 11-13.



Figure 11 Cloud chart of comprehensive stress results from finite element analysis of the tension rod after stress groove optimization

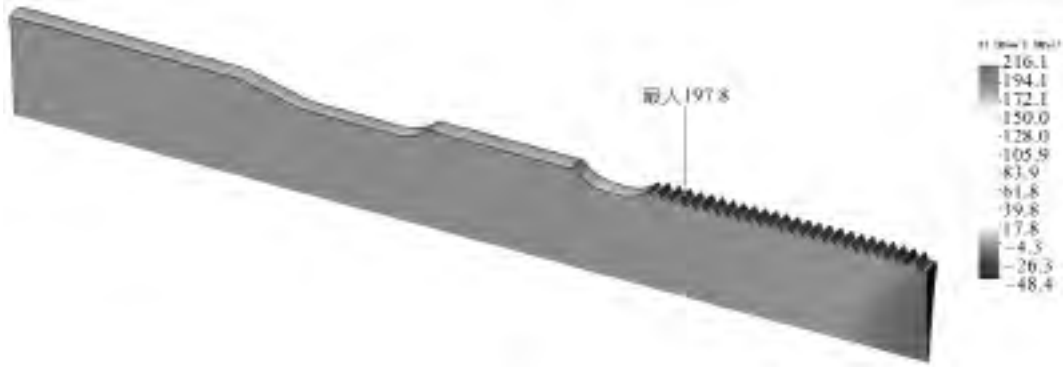


Figure 12 Cloud chart of the main stress P1 result of the finite element analysis of the tension rod after stress groove optimization

3 Conclusion and outlook

Through optimizing the design of the tension rod stress groove structure through finite element analysis, integrating multiple sets of dimensional parameters, and especially introducing the design of dual stress grooves, the comprehensive stress has decreased by 38% without changing the diameter of the tension rod or the nut structure. The main stress has decreased by approximately 40%, effectively reducing the probability of tension rod failure in injection molding machines. This is of great significance for the quality

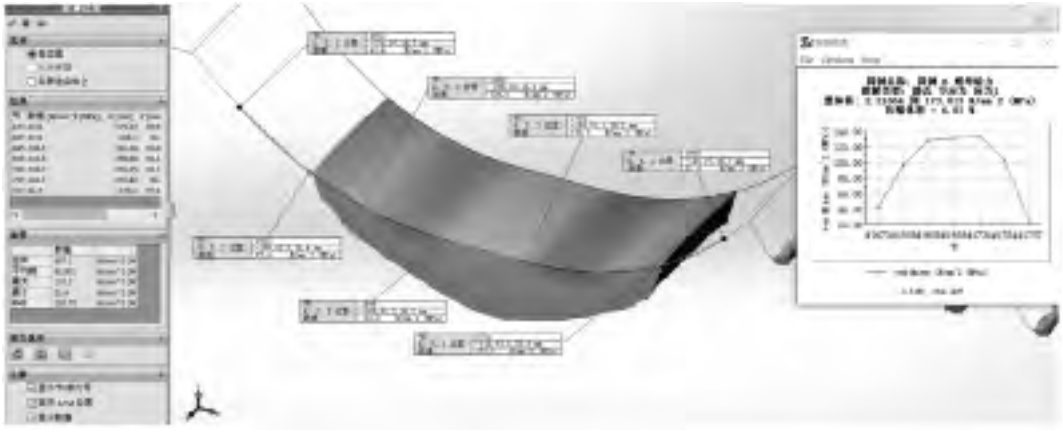


Figure 13 Stress detection results at the fillet and bottom diameter of the stress groove on the tie rod with width L1

and reliability of the tension rod. At the same time, optimizing the design of the tension rod stress groove has a significant impact on the stability of the clamping structure of the injection molding machine.

The optimized design method presented in this study can be applied not only in injection molding machines but also in die-casting machines, presses, forging machines, stamping machines, rubber machines, and other industries.

The research presented in this article has certain limitations. Factors such as the heat treatment of the tie rod, the processing technology of the tie rod thread, the material of the nut, and the connection structure between the tie rod and nut all have an impact on the load-bearing capacity of the tie rod. In the future, it is necessary to incorporate research on the influence of injection molding machine templates on the tie rod assembly, finite element analysis of the fatigue strength of the tie rod, and the development and application of high-strength tie rod materials. Furthermore, with the improvement of computer hardware performance and software, as well as the continuous investment of the country in the training of engineering personnel, it is believed that in the near future, finite element analysis of the entire machinery will be achieved, with increasingly higher precision and accuracy. The combination of theory and practice will become closer, and the machinery industry will become more prosperous.