



AS SEEN IN THE
INJECTION MOLDING
DIVISION NEWSLETTER

By Amos Shavit, Plastokit, Member of Rion Group

How to Determine Realistic Tolerances for Plastic Injection Molded Parts

The design of a product includes dimensions. Dimensions are required for functionality or correct fit in an assembly. It is impossible to produce identical parts; therefore, the designer defines tolerances for the design dimensions. These tolerances are to ensure that all dimensions fit the assembly requirements. Standards like DIN 16901 (and others) define general tolerances for different materials and different locations on the produced part. However, this is a general recommendation that cannot always be achieved in injection molding. The designer, most of the time, does not take into account the ability to produce the designed part.

This article discusses how to analyze the dimensions, tolerances, and ranges in injection molded parts in a more accurate and practical way to help the designer know in advance about production capability.



iD Additives™
Unmatched Technical Support for Innovative Products!



Eco-PRO 360

Cleans. Coats. Prevents.

A Better Option for Removing and Preventing Rust!

iD Eco-Pro 360 is a water-based, non-hazardous, ready to use rust remover, and very long term rust preventative.

BENEFITS

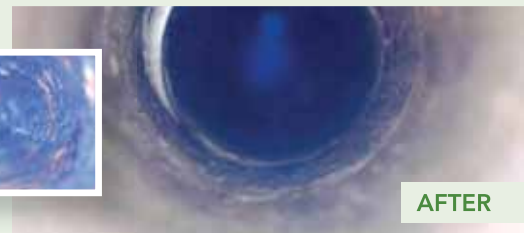
- Removes rust without etching
- Re-useable
- EPA tested and verified
- Made in the USA

BEFORE



AFTER

BEFORE



AFTER

BEFORE



AFTER

Contact iD Additives today for more information!

iD Additives, Inc • 512 W Burlington Ave., La Grange, IL 60525 • (708) 588-0081 • iDadditives.com

Definitions:

- **Tolerances** – are given by designer to designed dimensions
- **Deviation and ranges** - are the results of production
- **Range** - the difference between the smallest and the largest measurement of a dimension in a batch of produced parts

The injection molding process has its advantages because production is "in-mold". The cavities of the mold are produced from steel (hardened steel), and therefore, there is no dimension change in the mold. So, it is possible to deal with product dimensions by analyzing the shrinkage.

The following analysis is limited to:

- Dimension shrinkage only (not for distortion)
- No change in injection parameters

Simple Example

For illustrative purposes, let us start with a simple example (all dimensions in mm). **See Figure 1:**

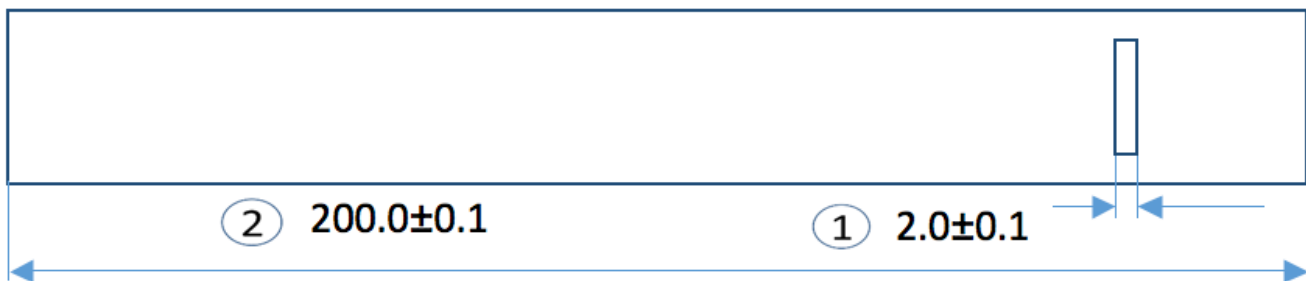


Figure 1: Sketch of a simple part.

Start with dimension no. 1 (2.0 ± 0.1). In order to achieve this dimension on the product, we add shrinkage for mold production. Assume that the shrinkage rate is 2%. Cavity dimension will be $2.0/0.98 = 2.0408$ mm, meaning: in order to meet the nominal dimension (2.0) the cavity dimension will be 2.0408mm. In the same part we have another dimension, no 2, (200.0 ± 0.1). For Dim 2, the mold dimension will be: $200.0/0.98 = 204.08$ mm. **See Figure 2.**

Mold dim (mm)	Shrinkage	Drawing dim (mm)	Dim no.
2.0408	2%	2.0 ± 0.1	1
204.08	2%	200.0 ± 0.1	2

Figure 2: Mold dimension to receive nominal part dimension.

How to Determine Realistic Tolerances for Plastic Injection Molded Parts

Dimensions of the cavity in the mold are fixed so shrinkage will be calculated from these dimensions.

On our simple part, each dimension can reach the tolerance limits. Now we can check what the shrinkage should be for each dimension in order to reach the limits. **See Figure 3.**

Percentage of tolerance from dimension	Shrinkage to min dim allowed	Min dim allowed (mm)	Shrinkage to max dim allowed	Max dim allowed (mm)	Mold (cavity) dim (mm)	Dim no
10%	6.9%	1.9	-2.9%	2.1	2.0408	1
0.1%	2.05%	199.9	1.95%	200.1	204.08	2

Figure 3: Shrinkage of the part that is needed to reach the tolerance limit.

Analysis of shrinkage to tolerance limits:

1. Dimension 1 – 2.0±0.1mm

- Shrinkage from mold size to maximum is impossible. Cavity dimension is smaller than the allowed maximum dimension on drawing (negative shrinkage).
- Shrinkage from mold size to minimum dimension is also impossible. There is no way that a material expected to shrink 2% will shrink 6.9%.
- The difference between minimum and maximum tolerance is 10% from dimension.
- This means that there is almost no chance that this dimension will be out of tolerance.**

2. Dimension 2 – 200.0±0.1mm

- The shrinkage from mold size to the maximum allowed dimension is 1.95%.
- The shrinkage from mold size to the minimum allowed dimension is 2.05%.
- The difference between maximum and minimum tolerance is 0.1% from dimension.
- This means that it will be much more difficult to produce this dimension within tolerance.**
- When the average shrinkage is not equal to 2% it will be almost impossible to produce within tolerance

Since these two dimensions are on the same part and in the same direction, the percentage of shrinkage of both of them will be equal to each other. Therefore, if dimension 200.0±0.1mm is maintained during production meaning the maximum range of production is 0.1% from dimension, the range of shrinkages of dimension 2.0±0.1mm will also be 0.1% (0.002mm). **See Figure 4.**

Range (mm)	Min dim for shrinkage of 2.05% (mm)	Shrinkage to min dim of dim no. 2	Max dim for shrinkage of 1.95% (mm)	Shrinkage to max dim of dim no. 2	Mold (cavity) dim (mm)	Dim no
0.002	1.999	2.05%	2.001	1.95%	2.0408	1
0.2	199.9	2.05%	200.1	1.95%	204.08	2

Figure 4: Calculation of shrinkage results when it is equal for both dimensions.

How to Determine Realistic Tolerances for Plastic Injection Molded Parts

This range of dimension 2.0 ± 0.1 mm (0.002mm) is very small for injection molded plastic parts and we do not have the ability to measure it or maintain it in production. Figure 4 relates to shrinkage of 2%. Later we will discuss different shrinkages. One conclusion is that there is a correlation between percentages of tolerance from drawing dimension, to the ability to produce the product. The higher the percentage, the easier the production and the opposite is also true. If the dimension 200.0 ± 0.1 mm is kept, there is no reason to measure the dimension 2.0 ± 0.1 mm. If, for example, the Quality Control Inspector measures this dimension and finds 2.08mm, it is easy to say that it is a mistake of measuring since the technology cannot produce this dimension.

Calculation of percentage of tolerance from drawing dimension (POT) is:

$$\frac{T}{D} \times 100 = POT$$

Where:

T – Tolerance

D – Dimension

POT – Percentage of Tolerance from dimension

The POT is defined by the part designer.

How do we know what the POT is that can be achieved? We have to connect it to the production capability.

Calculation of percentage of range from average (POR) is:

The production capability can be measured with the percentage of the range from average (POR).
Calculation of POR:

$$\frac{R}{A} \times 100 = POR$$

Where:

R – Range of results

A – Average of results

POR – Percentage of Range from average

POR is a result of production

- Selection of the right POR should be based on data collection from production
- POR depends on the quality of the injection process
- The more accurate the process, the lower the POR. For example:
 - ✓ Raw material. Should be uniform within batch and between batches
 - ✓ More accurate machines
 - ✓ More accurate molds with efficient and uniform cooling
 - ✓ Stable and repeatable injection process with big injection window
 - ✓ Using materials with low shrinkage
 - ✓ Uniform climate of injection molding facility (air-conditioned)
 - ✓ And so on

How to Determine Realistic Tolerances for Plastic Injection Molded Parts

- To begin the process, we can take a general expected number to all dimensions
- Good start for POR can be 0.2% to 0.3%
- Later on, after data collection, this number can be updated

Results that are more accurate can be achieved by taking measurements for some time and by calculating the percentage of range from dimension. As said before, it is possible to start with POR = 0.2% and change it later according to real results.

Now we can connect the expected percentage range of average (POR) with the POT. The POT enables us to rank the dimensions according to difficulty in production.

Real drawing analysis (based on a real part drawing – not shown here)

We used excel for this analysis (**Figure 5**):

Columns 1, 2, 3 and 4 will be filled with the dimension no. along with the specified dimension and tolerances from the drawing.

In column 5, the POT will be calculated for each dimension.

The expected POR (percentage of range from average) is shown in column 6. It is 0.2% for a starting point.

Now we rank the dimensions according to column 5 (POT) from smallest to largest.

1	2	3	4					5	6
Dim No	Nominal dim (mm)	Upper Tolerance (mm) +	Lower Tolerance (mm) -	Center Dim (mm)	Min (mm)	Max (mm)	Tolerance (mm)	POT (%)	Expected POR (%)
1	127.76	0.25	-0.25	127.76	127.51	128.01	0.50	0.39%	0.20%
2	85.47	0.25	-0.25	85.47	85.22	85.72	0.50	0.59%	0.20%
3	29.46	0.25	-0.25	29.46	29.21	29.71	0.50	1.70%	0.20%
5	9.80	0.13	-0.13	9.80	9.67	9.93	0.25	2.59%	0.20%
8	4.78	0.08	-0.08	4.78	4.70	4.86	0.16	3.35%	0.20%
6	6.05	0.13	-0.13	6.05	5.92	6.18	0.25	4.20%	0.20%
7	5.59	0.13	-0.13	5.59	5.46	5.72	0.25	4.54%	0.20%

Figure 5: Drawing dimensions, tolerances, POT and POR.

Now we produce a graph from data that will show the analysis of the drawing and tolerances with relation to our ability to produce the part. **See Figure 6.**

How to Determine Realistic Tolerances for Plastic Injection Molded Parts

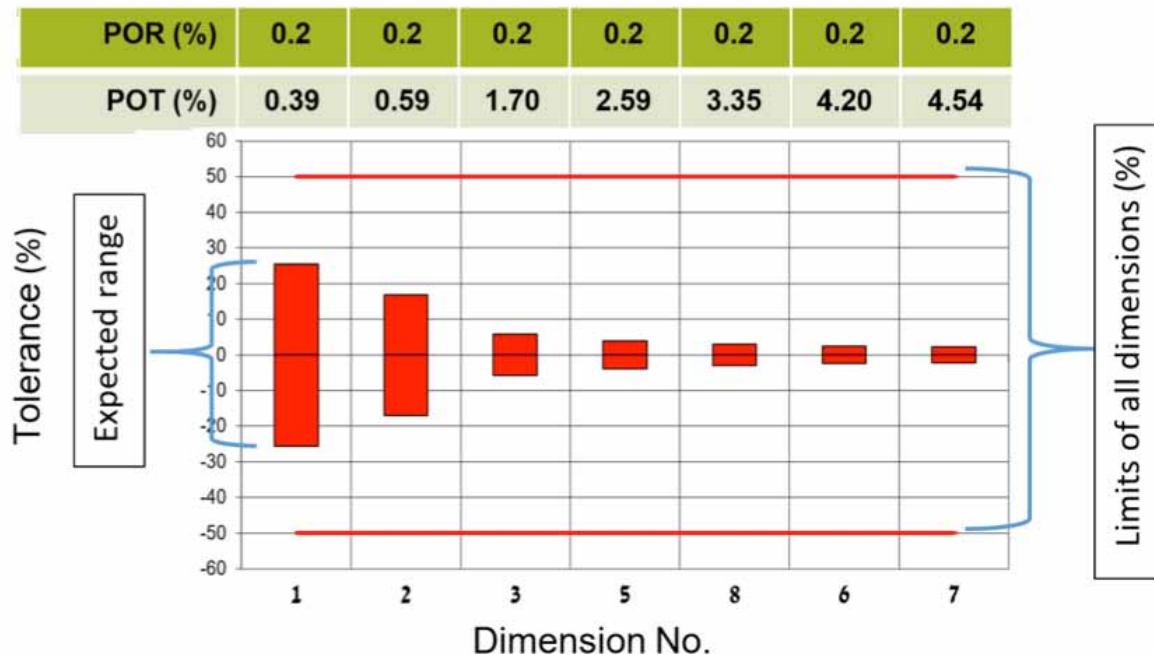


Figure 6: Analysis of drawing dimensions. Expected range of each dimension with relation to the tolerance borders, taking into account the general POR during production.

All dimensions in this graph are normalized to percentage. The upper and lower lines represent the maximum and minimum limits of each dimension, 50% above the nominal and -50% below the nominal.

The vertical red lines represent the distance between the expected ranges for each dimension with relation to dimensions limits.

The lower the POT, the more difficult it will be to produce the part (and vice-versa).
The higher the POR the more difficult it will be to produce the part (and vice-versa).

All the dimensions are ranked according to their difficulty to be produced. The graph in figure 6 shows that there is no problem to meet the tolerance requested since the columns are far from the limits.

Analysis of Measurement Results

Now we add the measurement results. **See Figure 7.**

How to Determine Realistic Tolerances for Plastic Injection Molded Parts

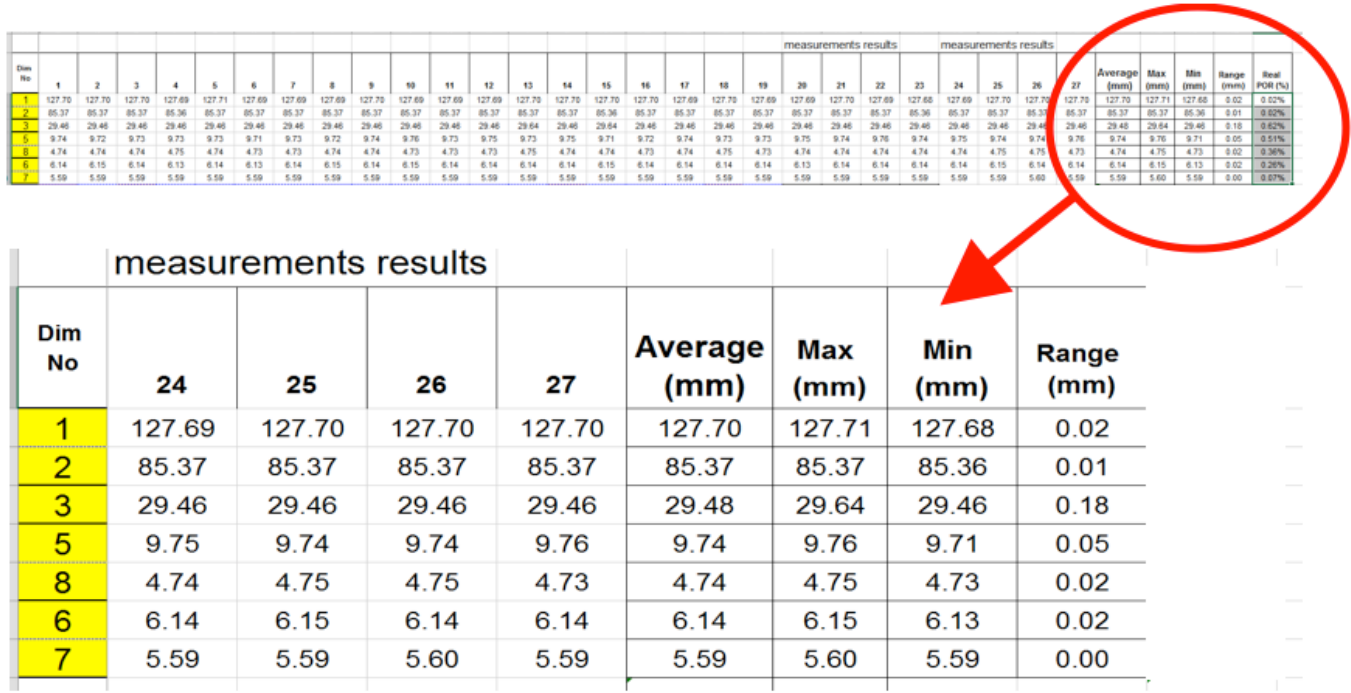


Figure 7: Measurement results and calculation of average, maximum and minimum to each dimension.

Columns 1 to 27 show measurement results (There is no limit to the number of measurements). At the end, there is a calculation of average, maximum and minimum for each dimension.

From this data, we produce a graph with all the results. **See Figure 8.**

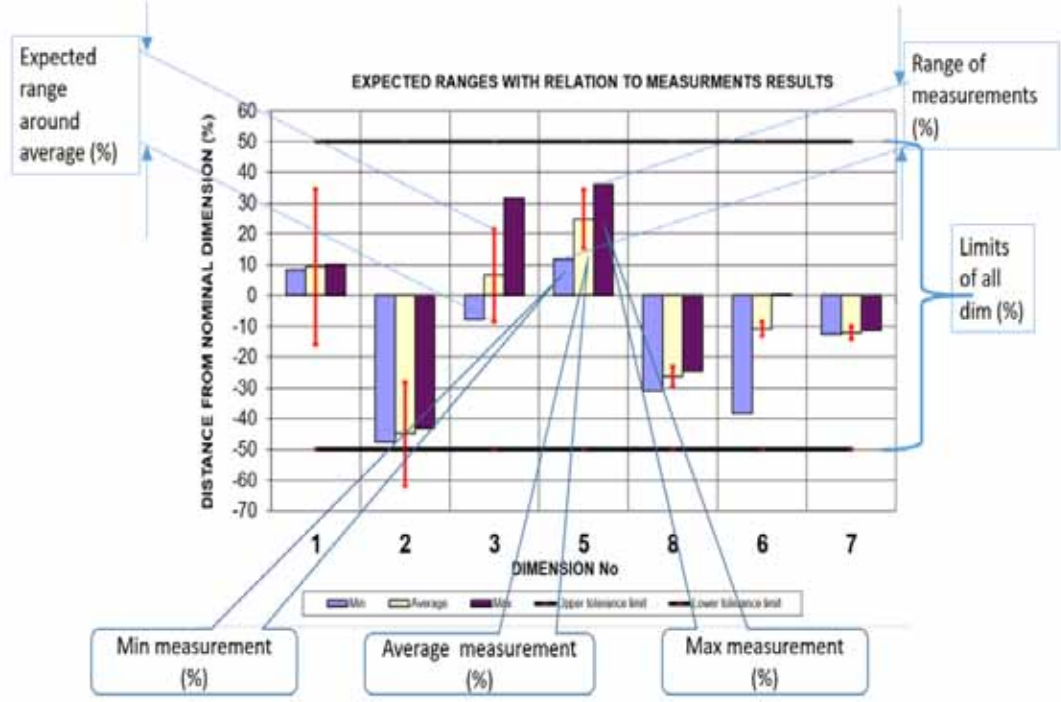


Figure 8: Results of actual measured data, average, min and max to each dimension and the expected POR.

Now results can be analyzed:

- The blue line is the minimum measured for each dimension
- The yellow line is the average calculated for each dimension
- The dark purple line is the maximum measured for each dimension
- The real shrinkage is different from 2% (if the real shrinkage would be 2% for all dimensions, the average of each dimension would be on the centerline). Above the centerline, the shrinkage is less than 2% and below the line is more than 2%.
- The expected POR (vertical red line) is around the average of each measured dimension.
- Measurements above zero shrink are less than anticipated (and are therefore larger).
- Measurements below zero shrink are more than anticipated (and are therefore smaller).
- **Measurements 1 and 2:** the actual range is much lower than expected (vertical red line is much longer than the difference between the dark purple line and the blue line).
- **Measurement 2:** measured dimension is within limits. The expected range shows that there is a possibility that the measurement will not meet the specification and can be too small. Expected range line (vertical red line), is crossing the lower limit.
- **Measurement 5, 8 and 7:** The actual range meets expectation. The expected range is between the minimum and the maximum.
- **Measurement 6:** the actual range is significantly greater than anticipated. i.e., the shrinkage significantly exceeded. Since no such range result is possible, it can be concluded that there is either a mistake in the measurement or data input.

Conclusions:

After collecting enough data, the prediction of POR will be much more accurate.

The data can be collected with relation to: materials, type of molds, size of molds, machines etc.

This suggested method enables us to:

- Analyze the feasibility of meeting the customer's part specification before entering the investment and commitment stages
- Be instrumental in forecasting which of the dimensions will be difficult to achieve.
- Determine the possibility of finding measurement errors.

This method can serve the following:

- For the designer of plastic parts
 - Define tolerances to dimension – feasibility of meeting the customer's requirements/specifications.
- The sub-contractor of injection molded parts before accepting order for new product.
 - Checking the possibility to meet drawing requirements and to select critical dimensions

Feature

- Mold maker
 - Checking the possibility of meeting customer requirements
- Mold test T1
 - Checking all drawing dimensions
 - Analysis of critical dimensions and ability to produce the parts
- Quality control
 - To give focus on critical dimensions that are difficult to produce
 - To find measurement mistakes

About Amos

Amos Shavit was a chief engineer and quality assurance manager for Naan Irrigation System in Israel. He received his MSc degree in Polymer Technology at Loughborough University, UK in 1992. He later worked as a consultant for Quality Assurance and Injection Molding. He also taught at colleges and businesses a course that he created about technology and quality in the injection molding process. He is currently working part time at Plastokit, Member of Rion group, an injection molding plant in Israel, and continues to teach injection molding in the industry.

Phone: +972-54-6741555 Mail: amos_sa@naan.org.il



Join **SPE**
SOCIETY OF
PLASTICS ENGINEERS



Students.....Young Professionals.....Professionals

Whether you're a plastics scientist or engineer, a business owner, marketing/sales expert, or any other professional in plastics, SPE membership can help you advance your knowledge and your career. The information you need to increase efficiency and productivity, develop your career, and add to your company's bottom line is literally right at your fingertips.

SPE's support of the development of plastics professionals begins at the moment they enter the industry as a student. Undergraduate and graduate students in plastics related fields of study can learn, study and build their professional networks.

For more information on membership benefits visit 4spe.org