Designation: D 3641 – 97

Standard Practice for
Injection Molding Test Specimens of Thermoplastic Molding and Extrusion Materials[1]

This standard is issued under the fixed designation D 3641; the number immediately following the designation indicates the year of original adoption or, in the case of revision, the year of last revision. A number in parentheses indicates the year of last reapproval. A superscript epsilon (ϵ) indicates an editorial change since the last revision or reapproval.

1. Scope *

1.1 This practice covers the general principles to be followed when injection molding test specimens of thermoplastic molding and extrusion materials. This practice is to be used to obtain uniformity in methods of describing the various steps of the injection molding process and to set up uniform methods of reporting these conditions. The exact conditions required to prepare adequate specimens will vary for each plastic material. These conditions should become a part of the specification for the material, or be agreed upon between the purchaser and the supplier.

1.2 The methodology presented assumes the use of reciprocating screw injection molding machines. Users of other types of machines will need to adapt this practice to their machines with appropriate precautions.

1.3 The values stated in SI units are to be regarded as the standard. The values given in parentheses are for information only.

1.4 *This standard does not purport to address all of the safety concerns, if any, associated with its use. It is the responsibility of the user of this standard to establish appropriate safety and health practices and determine the applicability of regulatory limitations prior to use.*

NOTE 1—This practice is equivalent to the following parts of ISO 294: Plastics—Injection Moulding of Test Specimens of Thermoplastic Materials[2]; 294-1: Part 1—General Principles and Multipurpose Test Specimens (ISO Type A Mould) and Bars (ISO Type B Mould)[2]; ISO 294-2: Part 2—Small Tensile Bars (ISO Type C Mould); ISO 294-3: Part 3—Plates (ISO Type D Moulds).

NOTE 2—Care should be taken in the design of the cavities for tensile test specimens to assure that the flat, parallel test area between the radii extending to the end tabs be of uniform dimensions and cross sectional area for the entire length. With some materials, lack of uniformity in dimensions or cross sectional area, or both, could lead to problems in testing. In these cases, it may be advantageous to slightly thicken the tabs and fillet (radii area) of the specimen.

2. Referenced Documents

2.1 *ASTM Standards:*

D 256 Test Methods for Impact Resistance of Plastics and Electrical Insulating Materials[3]
D 570 Test Method for Water Absorption of Plastics[3]
D 638 Test Method for Tensile Properties of Plastics[3]
D 648 Test Method for Deflection Temperature of Plastics Under Flexural Load[3]
D 788 Specification for Methacrylate Molding and Extrusion Compounds[3]
D 790 Test Methods for Flexural Properties of Unreinforced and Reinforced Plastics and Electrical Insulating Materials[3]
D 883 Terminology Relating to Plastics[3]
D 955 Test Method for Measuring Shrinkage from Mold Dimensions of Molded Plastics[3]
D 957 Practice for Determining Surface Temperature of Molds for Plastics[3]
D 3935 Specification for Polycarbonate (PC) Unfilled and Reinforced Materials[4]
D 4066 Specification for Nylon Injection and Extrusion Materials[4]
D 4101 Specification for Propylene Plastic Injection and Extrusion Materials[4]
D 4181 Specification for Acetal (POM) Molding and Extrusion Materials[4]
D 4507 Specification for Thermoplastic Polyester (TPES) Materials[4]
D 4549 Specification for Polystyrene Molding and Extrusion Materials (PS)[4]

2.2 *ISO Standards:*

ISO 3167 Plastics—Preparation and Use of Multipurpose Test Specimens[2]
ISO 294-1: Plastics—Injection Moulding of Test Specimens of Thermoplastic Materials—Part 1: General Principles and Multipurpose Test Specimens (ISO Type A Mould) and Bars (ISO Type B Mould)[2]
ISO 294-2: Plastics—Injection Moulding of Test Specimens of Thermoplastic Materials—Part 2: Small Tensile Bars (ISO Type C Mould)[2]

---

[1] This practice is under the jurisdiction of ASTM Committee D-20 on Plastics and is the direct responsibility of Subcommittee D20.09 on Specimen Preparation.

Current edition approved Dec. 10, 1997. Published May 1998. Originally published as D 3641 – 91. Last previous edition D 3641 – 93.

[2] Available from American National Standards Institute, 11 West 42nd St., 13th Floor, New York, NY 10036.

[3] *Annual Book of ASTM Standards*, Vol 08.01.

[4] *Annual Book of ASTM Standards*, Vol 08.03.

***A Summary of Changes section appears at the end of this standard.**

ISO 294-3: Plastics—Injection Moulding of Test Specimens of Thermoplastic Materials—Part 3: Plates (ISO Type D Moulds)[2]

3. Terminology

3.1 *Definitions*—Definitions of terms applying to this practice appear in Terminology D 883.

3.2 *Definitions of Terms Specific to This Standard:*

3.2.1 *average injection velocity*, *n*—the mean value of the velocity of the molten plastic flow front within a cavity during the injection time that is calculated from the shot volume and injection time. (See Terminology D 883.)

3.2.1.1 *Discussion*—The average injection velocity is calculated as follows:

$$V_{av} = \frac{V_s}{T_i \times A_c \times n}$$

where:

V_{av} = average injection velocity, mm/s,
V_s = shot volume, mm^3,
T_i = injection time, s,
A_c = cross section of the cavity, mm^2, and
n = number of cavities.

The calculation is valid for molds containing a single cavity or those containing identical multi-specimen cavities only and not for family molds.

3.2.2 *cross section of the cavity*, *n*—in a mold for test specimens, the area of a planar section perpendicular to the flow pattern during filling of the mold that forms the critical portion of the test specimen.

3.2.3 *flash*, *n*—thin fin of material formed at the parting line of a part during molding, caused by unintentional opening of the mold or by defective mating surfaces.

3.2.4 *switchover point*, *n*—that point in the injection stage of the injection molding cycle when the control or level of the forwarding force on the screw is switched from that used during injection to that used during pack/hold.

3.2.5 *velocity-pressure transfer point (VPT)*, *n*—that point in the injection stage of the injection molding cycle as defined by pressure, position, or time when the rate of ram travel is switched from speed control to pressure control.

4. Summary of Practice

4.1 Many factors in the injection molding process can have an influence on the character of the moldings and the numerical values of test results. Among these are geometry, size, and temperature conditions of the heating chamber, pressures and speeds used, size, shape, and length of runners and gates, mold temperature and its uniformity, cavity surface finish, and timing cycles used along with the method of sequencing from stage to stage in the process. It is sometimes necessary to pretreat materials before molding. For materials that absorb water this may mean drying under prescribed conditions. This practice attempts to control some of these variables, nullify others, and report those that are necessary to the obtaining of reproducible specimens. Definite stock and mold temperatures, based on the relevant material specifications or the material supplier's recommendations or past experience, and measured by standard techniques, are used for the molding process. By a sequence of operations the pressures, timing settings, and mode of control are established on the basis of their effects upon the molded part itself rather than upon any universal setpoints.

5. Significance and Use

5.1 It is well known that plastic test specimens molded under different conditions can have significantly different properties. This practice is designed to minimize those differences by establishing operating protocols without being unnecessarily restrictive.

5.2 Always refer to the ASTM material specification or ISO designation for the material for recommended molding conditions. If not available, consult the material supplier.

5.3 This practice requires the use of adequate quantities of plastic material to find desirable operating conditions and to make the desired test specimens.

6. Apparatus

6.1 *Injection Molding Machine*—The machine selected for use must be equipped with appropriate devices for the control and measurement of hydraulic pressure, all relevant temperatures, and the timing of certain cycle elements. Additional devices to monitor cavity pressure and ram position and velocity are very useful in the effective control of the injection molding cycle. The capacity of the machine should be such that the total shot weight (specimens plus sprue and runners) is 20 to 80 % of rated capacity. The injection machine must be capable of maintaining the proper injection velocity range if specified in the material standard.

NOTE 3—Heat-sensitive materials may require using the high end of the 30 to 80 % range in order to minimize residence time of the melt in the barrel.

6.1.1 *Control System*—The various control systems should be able to maintain the operating parameters of the injection molding process from cycle to cycle within the following limits:

plastic melt, or stock temperature	±3°C
mold temperature	±3°C, ≤80°C ±5°C, >80°C
injection pressure	±2 %
hold pressure	±5 %
injection time	±0.1 s
hold time	±5 %
shot weight	±1 %

Suitable means of monitoring these parameters to ensure control within the above limits should be present.

NOTE 4—The mold temperature may be measured using a calibrated surface pyrometer and the technique described in Practice D 957.

6.1.2 *Screw*—The design of the screw will be determined by the material being molded. A key criterion of screw design is to provide a melt that is as uniform as possible with respect to composition, temperature, and viscosity.

6.1.3 *Clamp*—The clamping force of the machine shall be high enough to prevent flashing at all operating conditions.

NOTE 5—The minimum clamp force required is the product of the highest possible cavity pressure and the projected surface area of the cavities (and runners). A force greater than this minimum will be required to prevent flashing.

6.2 *Mold*—The design of the mold is one of the more critical variables affecting specimen properties. Optimum reproducibility requires that identical molds be used by parties attempting to obtain comparable results. However, in the absence of identical molds, adherence to certain features of design will help to minimize differences between results obtained by different parties. It has been found that the use of unitized mold bases with interchangeable mold plates and gate inserts can provide a great deal of flexibility and provide rapid transitions between the moldings of different specimen configurations. (See Annex A1.)

6.2.1 *Cavity Layout*—Multi-cavity molds with identical cavities are recommended. The cavity layout should be such that there is a uniform and symmetrical distribution of specimen surface area on the overall mold surface. The use of single cavity molds is discouraged. For large tensile test specimens and multipurpose bars, a two cavity "Z" or "T" layout is acceptable. For small tensile test specimens and bars, a four cavity double "T" layout is recommended. Other specimens have their unique cavity layout. (See Annex A1.)

NOTE 6—ISO 294-1 states that the "Z" cavity runner layout is preferred over the "T" cavity runner layout.

NOTE 7—Family molds designed to produce more than one part configuration with each shot are not recommended. If molds of this nature are used, consideration should be taken in the design to ensure that constant and uniform filling velocities are achieved in all cavities. Additionally it is cautioned that the comparability of data obtained on specimens molded in this manner may be limited not only to a specific polymer type but also to specific rheological characteristics.

6.2.2 *Runners*—Runners may be of the full-round type cut into both halves of the mold or of the trapezoidal type cut into only one of the mold halves. They should be a minimum of 5 mm (0.2 in.) in diameter or of equivalent cross-sectional area if trapezoidal. A symmetrical cavity layout will permit identical runner systems to be used for each cavity and thus facilitate uniform filling of all cavities with all materials under all conditions. Runner draft angles for trapezoidal runners should be from 10 to 30°C. The diameter of the sprue shall be a minimum of 4 mm on the nozzle side.

NOTE 8—If family molds with two or more identical specimen cavities or non-identical runner systems or if multi-cavity molds with non-identical runner systems are used, specimens from such cavities are to be identified and should not be commingled for testing unless it has been demonstrated that there are no statistically significant differences in test results between the cavities.

6.2.3 *Gates*—Unless otherwise stated for specific specimens, or material specifications, the gate depth should be at least two-thirds the depth of the bar-type cavities and the gate width should preferably be equal to the width of the bar-type cavities but no less than two thirds the width. Gates should be as short as possible with a maximum length of 3 mm (0.12 in.). Such large gates tend to give parts whose physical properties are less sensitive to varying molding conditions than smaller ones. However, many existing test methods call for somewhat smaller gates such as some of those listed in Table 1. Also certain materials may require small gates to promote shear thinning so that mold cavities can be filled. Details of such gates shall become a part of the injection molding report.

6.2.4 *Cavities*—Machining tolerances of the cavity will depend on the material to be molded and on the tolerances allowed in the specific test methods. Dimensioning of cavities with respect to anticipated shrinkage will result in cavity variations from molder to molder. It also requires different molds for every material to be molded. It is preferred to machine the cavity to the nominal dimensions of the specimen and to adjust the dimensions only when shrinkage leads to a specimen that is out of specification for the desired test method. Draft angles in the sidewalls of the cavity will probably be needed to facilitate part ejection but shall not be greater than 1° except in the shoulder of the multipurpose test specimen (ISO 3167) that shall be not greater than 2°. A maximum draft angle of ½ ° is preferred for all areas. All interior mold surfaces should be finished to Society of Plastics Industry-Society of Plastics Engineers (SPI-SPE) No. A-3 or better. If cavity identification is required, this should be located outside of the test area. It is recommended that the end of an ejector pin be used to incorporate an identifying symbol rather than the cavity surface.

TABLE 1 Recommended Gate Sizes for Common Test Specimens

Specimen	Gate Size, mm Depth Width	Test Method
Type I tension specimen	3.2 by 9.6	D 638
3.2 by 12.7 by 127 mm bar	3.2 by 6.4	D 256, D648, D790
6.4 by 12.7 by 127 mm bar	6.4 by 6.4	D 256, D648, D790
3.2 by 51 mm disk	3.2 by 3.2	D 570
3.2 by 102 mm disk	3.2 by 12.7	D 955
Multipurpose test specimen	2.5 by 14 (min)	ISO 3167

6.2.5 *Ejector Pins*—Ejector pins shall be located where necessary, but not in the test area of the specimen. For tensile test specimens, it is recommended that the ejector pins be located at the wide tab ends (shoulders). For universal bars, it is recommended that the ejector pins be located at the "dead" end of the bar and outside of the central 20 mm length of the ISO mold type B (80 mm bar). For plates, the ejector pins should be outside the central area of 50 mm diameter.

6.2.6 *Cavity Pressure Transducers*—Pressure transducers may be mounted in conjunction with an ejector pin or directly into the cavity, coplanar to the mold surface. These may be used to monitor or control the mold filling characteristics. In the case of plates, the pressure transducer is mandatory for measurement of molding shrinkage only and the mounting shall be as shown for ISO mold type D in Annex A1. Transducers may also be located in the main runner but this location will not provide suitable data for monitoring or controlling cavity flow.

6.2.7 *Molding Cooling*—Coolant channels shall be designed so as to limit point-to-point surface temperature differences to a maximum of 5°C. The channels should be located in such a way that each of the cavities is cooled in an identical manner. Although this may not be possible with non-symmetrical family molds, it is a design criterion that should be followed as closely as possible.

NOTE 9—Fittings used to connect the coolant channel openings to the recirculating system may be of the self-sealing quick-disconnect type. For maximum cooling efficiency the flow of coolant through the channels should be turbulent, as indicated by a Reynolds number >5000.

7. Procedure

7.1 *Consistency of Test Specimens*— Consistent test specimens can only be obtained when operating on an automatic cycle with adequate control of the parameters listed in 6.1.1.

7.2 *Feed*—Condition plastic material prior to molding as required in the relevant material standard or according to the material supplier's recommendation if no standard covers the material. In the case of extreme moisture sensitivity, maintain drying conditions in the molding machine feed hopper itself. Avoid exposing material to an atmosphere with temperatures significantly below ambient temperature to prevent condensation of moisture onto plastic material.

7.3 *Setting Molding Conditions*—Set molding conditions as specified in the relevant material standard, or according to the material supplier recommendations, if available. The molding conditions that are critical are melt temperature, mold temperature and average injection velocity. Others that may be specified include first and second stage injection pressure, and screw back pressure; first and second stage injection times; cure/cool, mold open, melt decompression and total cycle times; screw rotation speed and cushion/pad interval.

7.3.1 *Melt Temperature*—Choose a melt, or stock temperature based on the relevant material standard, or the recommendations of the material supplier, or best available information. Adjust the barrel temperature controller setpoints in order to obtain the desired melt temperature. These settings will typically be lower than the aim melt temperature. Measure this temperature on cycle after thermal equilibrium has been reached by actually taking the temperature of a free shot (minimum volume of 30 ccm) with a needle-type pyrometer having an accuracy of ±2.0°C (3.8°F). Move the needle about constantly in the plastic mass and make a sufficient number of measurements to establish a reliable average. Preheating of the needle to near the measured temperature will allow faster, and thus more reliable measurements to be made. Perform frequent checks. The melt temperature may alternatively be measured by means of a suitable temperature sensor in the molten plastic flow stream provided the values obtained are the same as those by the free shot method.

NOTE 10—Under a fixed set of operating conditions of cycle and shot size there will generally be a constant difference between the observed plastic temperature and the cylinder-wall temperature or the setpoint temperature. This difference will vary as conditions are changed.

7.3.2 *Mold Temperature*—Choose a mold temperature based on the relevant material standard or the supplier's recommendation. The temperature should be measured with a surface-type pyrometer to an accuracy of ±2.0°C (3.8°F) after the system has obtained thermal equilibrium and immediately after opening the mold and ejecting the part. Carry out measurements of the mold cavity surface temperature at several points on both the moveable and stationary mold halves. The molding process shall be recycled a minimum of ten cycles between each set of multiple readings. Calculate the mold temperature as an average of these readings.

7.3.3 *Average Injection Velocity*—Unless otherwise specified, an average injection velocity of 200 ± 100 mm/s is suitable for most materials in preparing tensile specimens and bars. For other specimens, average injection velocity is chosen to give similar fill times as with tensile and bar specimens.

7.3.4 *Molding Pressures*—Most injection molding machines use hydraulic pressure as the motive force for the various machine movements. In different machines valves are used to control the level of pressure, the flow rate of fluid, the length of time at a given pressure, or combinations of these parameters. Filling of the mold is typically done with a high pressure (first stage) in order to achieve the desired short fill times. However, since this high pressure may cause flashing of the mold after the cavity is filled, and since release of the pressure after filling would allow polymer to flow out of the mold, a usually lower second stage, or hold pressure, is begun when the mold is essentially filled. The point in the cycle when this transition is made is called the "switchover point" or the velocity-pressure transfer point in those machines where the first stage is speed controlled. Control of these functions is done using a variety of means involving timers, position sensors, pressure transducers, or combinations thereof.

7.3.4.1 *First Stage Injection Pressure*— After choosing a melt and mold temperature from the relevant material specification or by other means the first stage injection pressure may be determined by the following procedure. Set the second stage pressure to zero and incrementally increase the first stage pressure from an obviously low value where long fill times and incompletely filled cavities occur until the fill time is reduced to its target value and the cavities are filled. The readouts on the timer, position sensor, or pressure transducer should be noted as control parameters for the final cycle. Maintain as constant a screw forward speed during the injection period as possible.

7.3.4.2 *Second Stage Injection Pressure*— Having chosen the melt and mold temperatures, the first stage injection pressure and its means of control, set the second stage pressure by adjusting the second stage pressure upward until a satisfactory part is produced. This is indicated mainly by surface appearance, that is, lack of sink marks, voids, flash, etc. Second stage pressure should be maintained constant until polymer in the gate of the cavity is sufficiently cool so that flow through the gate is no longer possible or the "gate freezes." This can be determined by increasing the setpoint of the timer until a constant part weight is obtained.

7.3.4.3 *Back Pressure*—After the second stage time has expired the screw will begin to rotate to plasticate the polymer for the next injection cycle. As melted polymer is forwarded to the barrel ahead of the screw the screw will backup against the imposed back pressure. High back pressure will cause excessive working of the polymer, high melt temperatures, and drooling of polymer from the nozzle. It will also cause breakdown of any fibrous fillers. Low back pressure may cause unmelted polymer or nonuniform polymer temperatures or inadequately mixed polymers where additives are used. Choose a back pressure that avoids the difficulties of the extremes described.

7.3.5 *Molding Cycle*—Injection-molding machines of different types and produced by different manufacturers have different means of controlling the molding cycle. Fig. 1 illustrates the interplay of the different components of the cycle. Certain ones of these will be controlled by various means on the different machines. Although the cooling and