SOLIDWORKS® 2022 Advanced Techniques

Mastering Parts, Surfaces, Sheet Metal, SimulationXpress, Top Down Assemblies, Core & Cavity Molds

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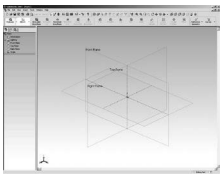

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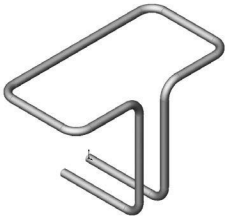

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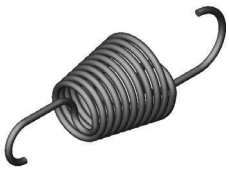

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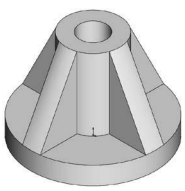

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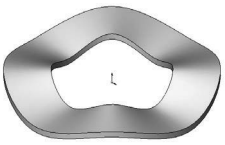

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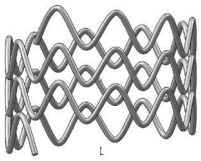

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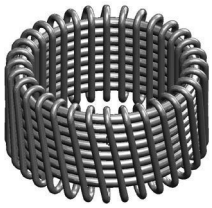

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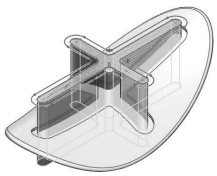

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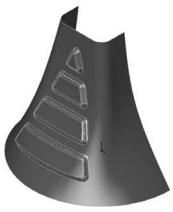

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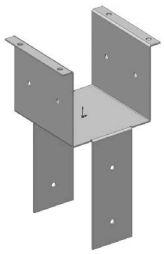

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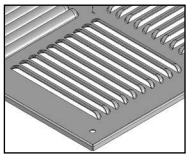

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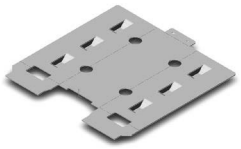

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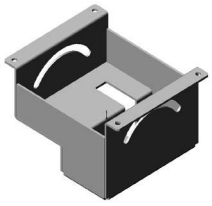

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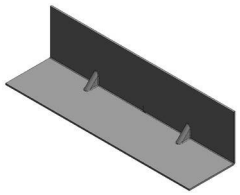

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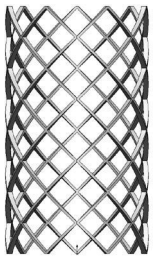

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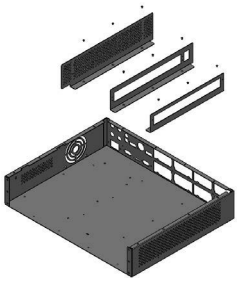

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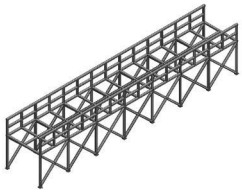

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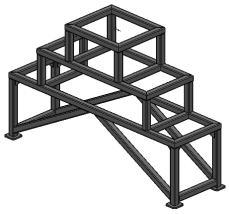

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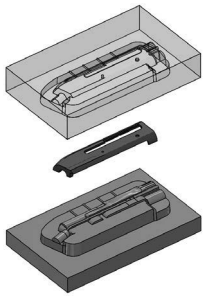

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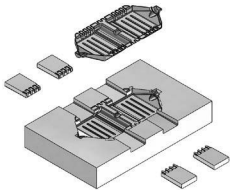

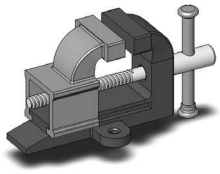

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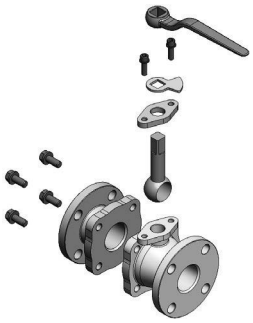

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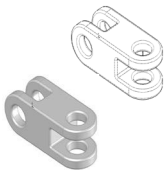

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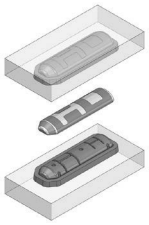

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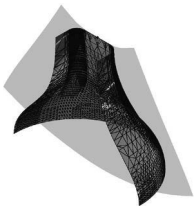

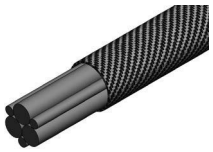

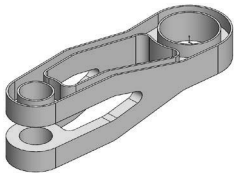

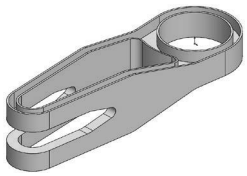
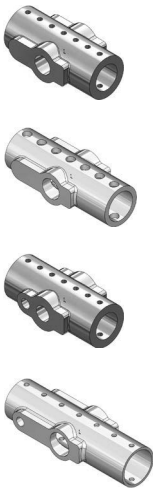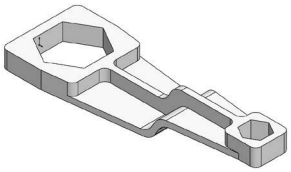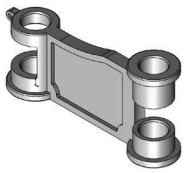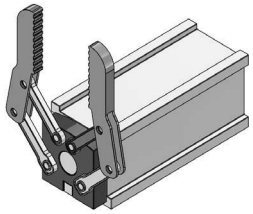
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Quick Reference Guide to SOLIDWORKS 2022 Command Icons and Toolbars.