SOLIDWORKS® 2020 Intermediate Skills

Expanding on Solids, Surfaces, Multibodies, Configurations, Drawings, Sheet Metal and Assemblies

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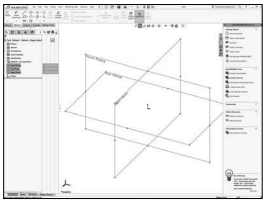

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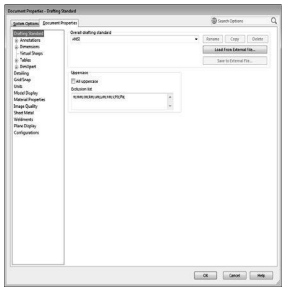

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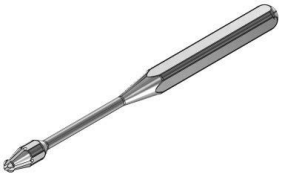

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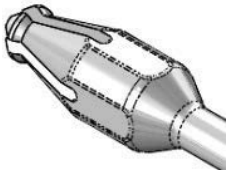

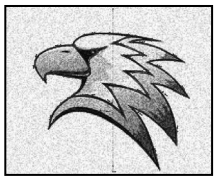

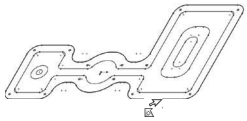

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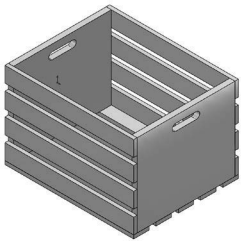

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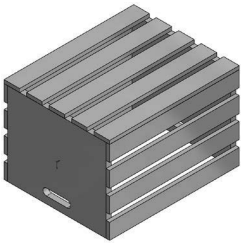

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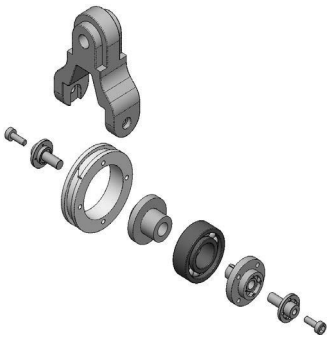

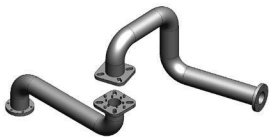

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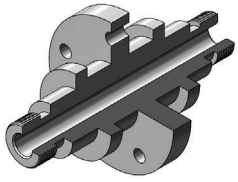

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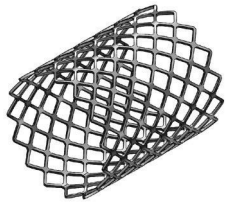

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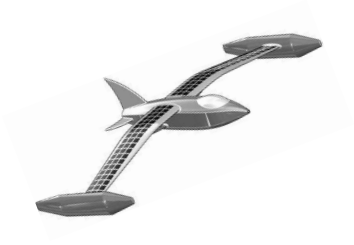

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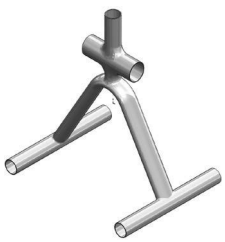

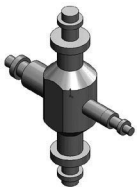

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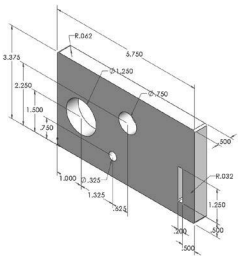

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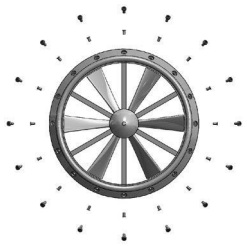

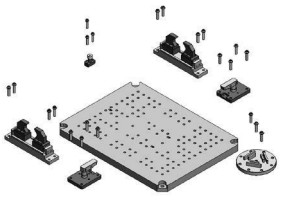

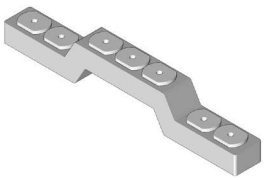

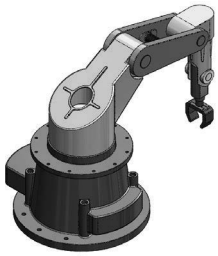

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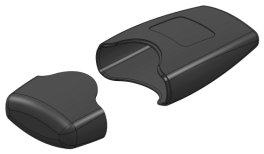

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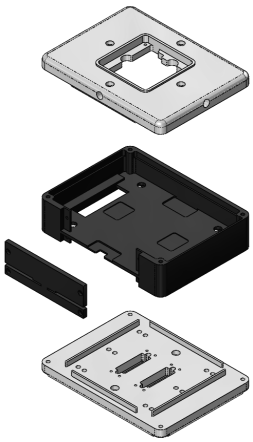

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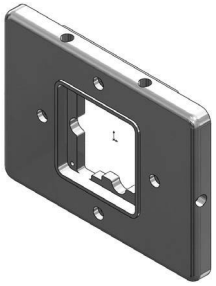

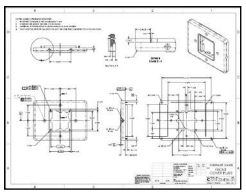

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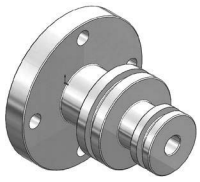

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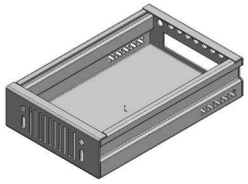

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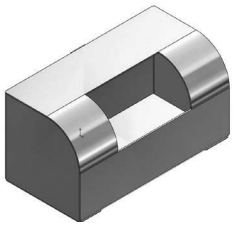

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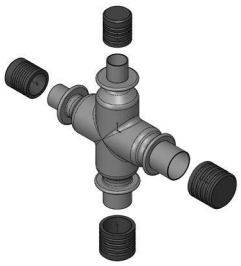

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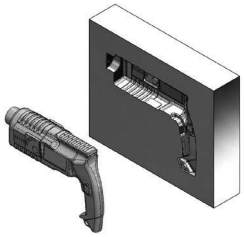

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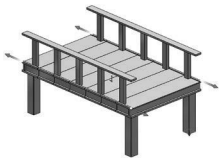

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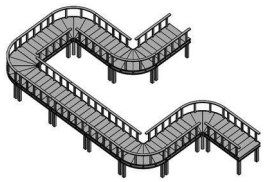

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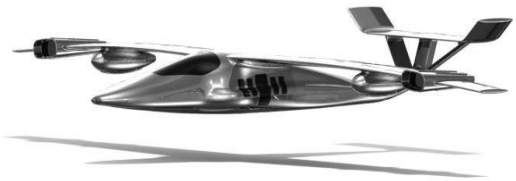
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