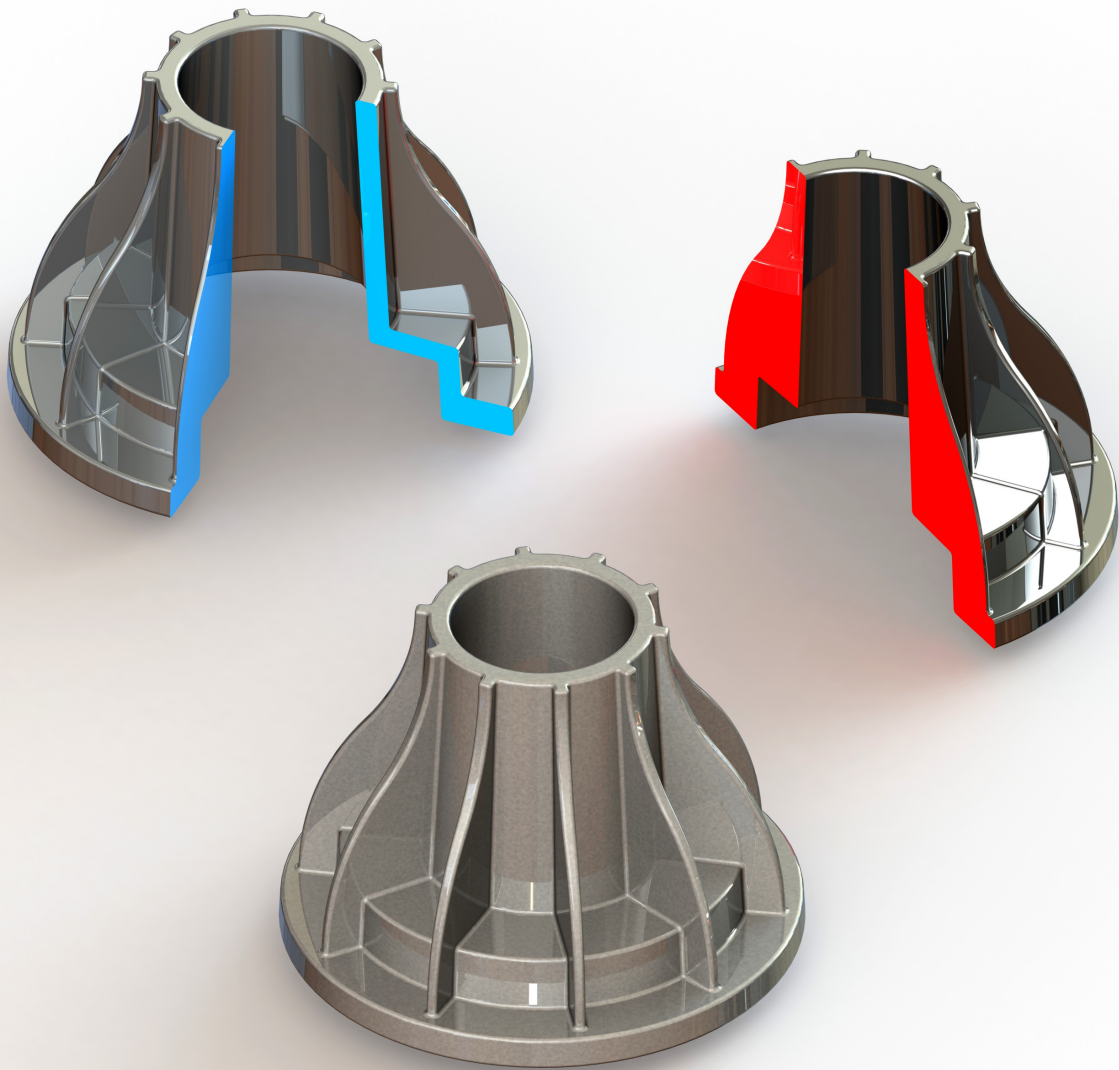


SOLIDWORKS® 2021

Basic Tools

Getting Started with Parts,
Assemblies and Drawings



Paul Tran CSWE, CSWI

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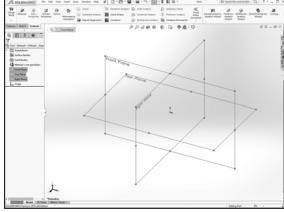
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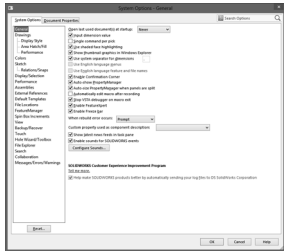
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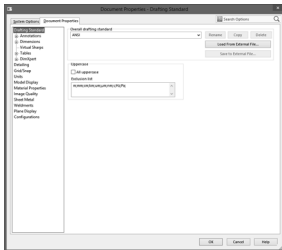
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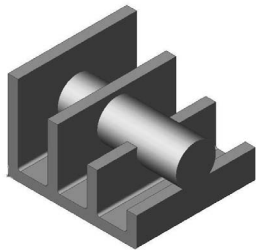
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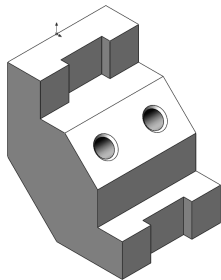
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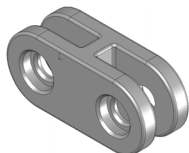
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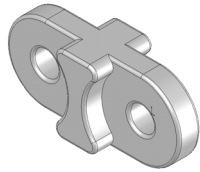
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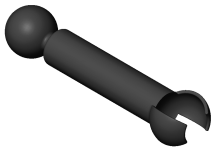
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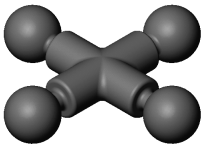
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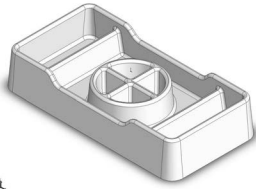
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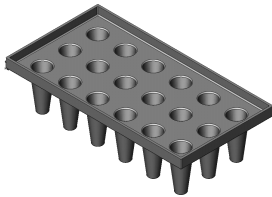


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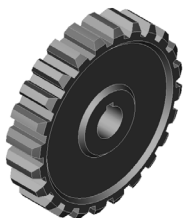
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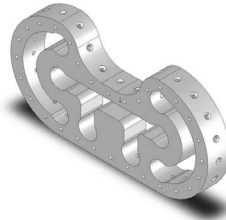
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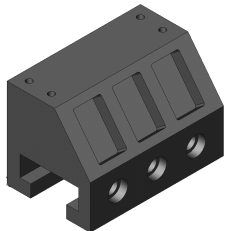


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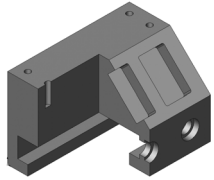
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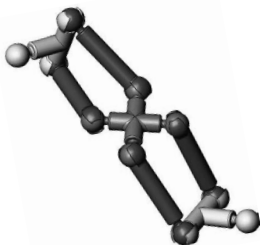
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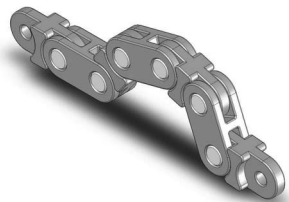
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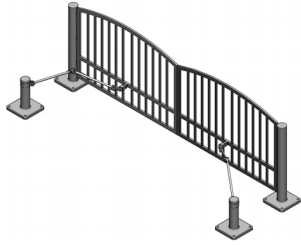
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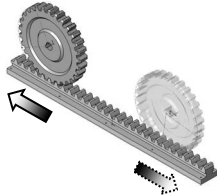


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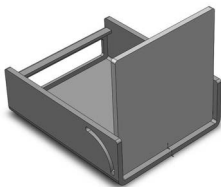
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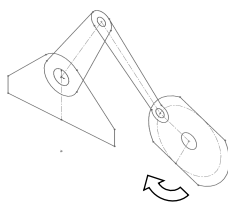
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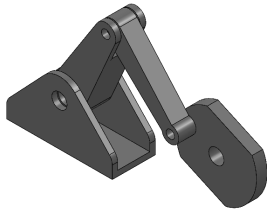
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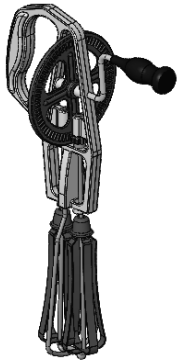
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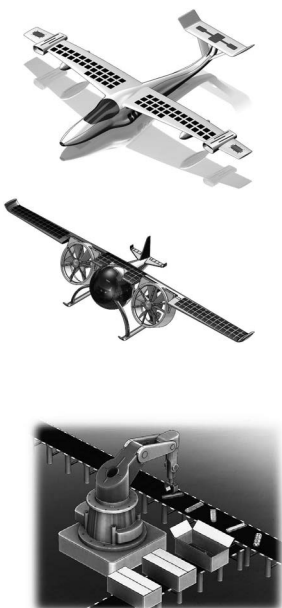


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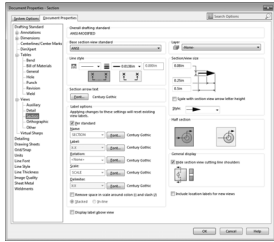
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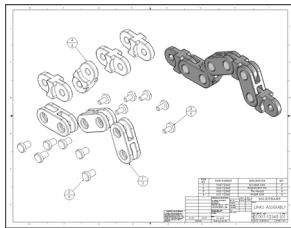
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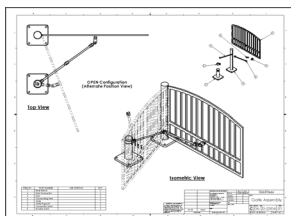


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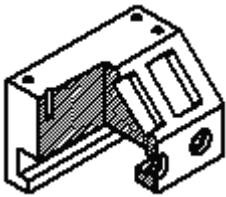
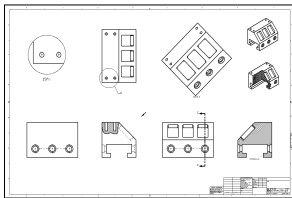


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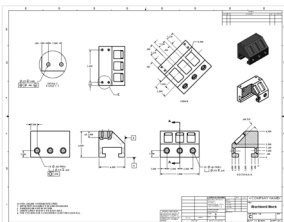
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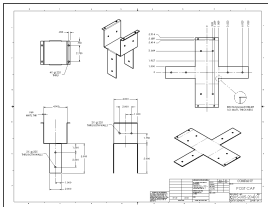
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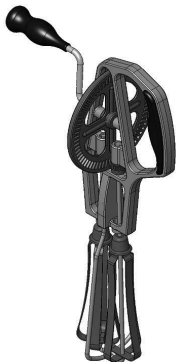


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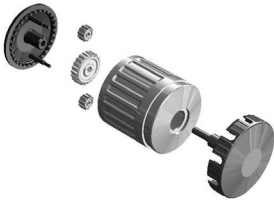
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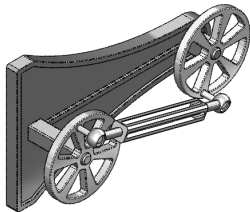
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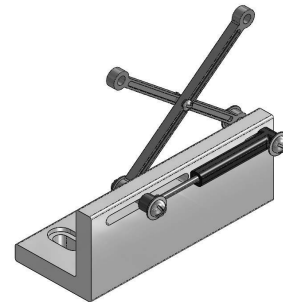
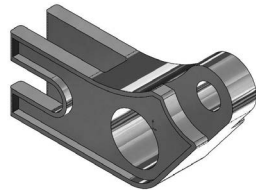
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